

EnRoute 4

User's Guide

Version 4.1

4/09



© Copyright 2004-2008
ALL RIGHTS RESERVED



Table of Contents

TABLE OF CONTENTS	III
1. WELCOME	1
WELCOME TO ENROUTE	1
WHAT'S NEW IN ENROUTE	1
RECOMMENDED SYSTEM REQUIREMENTS	3
HARDWARE KEY	3
INSTALLING THE SOFTWARE	4
ENROUTE 4 SUPPORT SERVICES	5
2. GETTING STARTED	7
WORKFLOW	7
BASIC ELEMENTS OF ENROUTE	7
WORKING WITH VIEWS	8
MENUS	8
TOOLBARS	8
CHANGING UNITS	9
STATUS LINE	9
USING SCROLL BARS	10
PRECISION INPUT CENTER	10
USING SNAPS	11
USING GUIDELINES	12
EDIT GUIDELINES DIALOG	12
CHANGING THE VIEW	14
ZOOM COMMANDS	15
REDRAW	15
VIEW SETUP	15
USING THE UNDO AND REDO FEATURES	16
WORKING WITH LAYERS	16
CHANGING A CONTOUR'S LAYER	18
USING HELP	18
SETTING PREFERENCES	19
EXITING THE SOFTWARE	26
3. TOOLBARS AND COMMAND ICONS	27
OTHER COMMAND ICONS	44
4. WORKING WITH FILES	51
CREATING NEW FILES	51
USING TEMPLATES	53
OPENING A DESIGN	53
SAVING A DESIGN	53
CLOSING A DESIGN	54
IMPORTING A DESIGN	54
EXPORTING A DESIGN	54

PRINTING	55
CALCULATOR	55
SNAPSHOT	56
MEASURE	56
5. CREATING CONTOURS	59
LINE, POLYARC OR BEZIER CURVE	59
RECTANGLE	64
CIRCLE	65
ARC.....	67
ELLIPSE	68
POLYGON	69
TEXT	70
DIMENSIONS	73
6. CONTOUR EDITING	77
EDITING POINTS ON A CONTOUR	77
EXTENDING CONTOURS.....	79
TRIMMING CONTOURS	81
FILLETING CONTOURS	83
JOINING CONTOURS	85
OFFSET CONTOURS	86
PARTIAL OFFSET CONTOURS.....	86
MERGING SELECTED OPEN CONTOURS.....	87
EXPLODE SELECTED CONTOURS.....	88
CONVERTING CONTOURS TO ARCS	88
REVERSE OPEN CONTOURS	88
CREATING MULTIPLE COPIES OF A CONTOUR	89
VECTORIZING BITMAPS.....	94
CLEANING UP CONTOURS.....	95
7. ARRANGING CONTOURS.....	97
SELECTING CONTOURS	97
CUTTING, COPYING AND PASTING CONTOURS	97
DELETING CONTOURS	98
GROUPING AND UNGROUPING CONTOURS.....	99
MOVING, SCALING, AND ROTATING OBJECTS.....	99
MOVE TOOL	103
SCALE TOOL	104
ROTATE TOOL.....	108
DISTORT CONTOURS	110
MIRRORING A CONTOUR	111
ALIGNING CONTOURS	114
WELDING CONTOURS	115
USING THE CUT BY LINE TOOL	118
NESTING CONTOURS.....	119
8. WORKING WITH TOOLPATHS	125

OVERVIEW	125
CREATING ROUTING OFFSET TOOLPATHS	126
CREATING OPEN CONTOUR OFFSET TOOLPATHS	136
CREATING HATCH FILL TOOLPATHS	138
CREATING ISLAND FILL TOOLPATHS	143
SPIRAL FILL TOOLPATHS	146
DRILL TOOLS	147
CREATING SLOT TOOLPATHS	156
CREATING AN ENGRAVE TOOLPATH	157
CREATING PYRAMID TOOLPATHS	159
EDITING TOOLPATHS	161
EDITING THE TOOLPATH PLAN	162
EDITING ENTRY/EXIT POINTS	163
EDITING BRIDGES	165
DELETING TOOLPATHS	167
9. PREVIEWING OUTPUT	169
SIMULATING OUTPUT IN 2D	169
SIMULATING OUTPUT USING AN ORTHOGRAPHIC VIEW	171
CREATE RENDERED VIEW OF TOOLPATHS	172
10. CONFIGURING THE SOFTWARE	175
CONFIGURING MACHINE DRIVERS	175
EDITING THE MATERIAL LIBRARY	181
EDITING THE TOOL LIBRARY	182
11. OUTPUT TO MACHINE	187
SETTING THE TOOLPATH ORDER	187
GENERATING OUTPUT	188
USING OUTPUT CONTROL CENTER TO MANAGE OUTPUT	193
12. 3D SURFACES AND 3D TOOLPATHS	197
INTRODUCTION	197
3D SURFACE APPLICATIONS	197
CREATING 3D SURFACES	198
CREATING OUTPUT FOR 3D SURFACES	198
13. CREATING RELIEFS	199
CREATING A RELIEF	199
RELIEF SHAPE	199
RELIEF OPTIONS	200
APPLICATION METHOD	202
RELIEF PARAMETERS	205
USING THE RELIEF DIALOG	208
14. REVOLUTIONS AND EXTRUSIONS	213
INTRODUCTION	213
REVOLVE	213

SPIN	219
EXTRUDE	230
SWEEP TWO RAILS.....	239
15. CHAMFERING	247
INTRODUCTION.....	247
STANDARD CHAMFER	247
CHAMFER CENTERLINE.....	254
BAROQUE CARVE CHAMFER	258
16. USING 3D MESHES	261
INTRODUCTION.....	261
USING 3D MESHES FROM OTHER APPLICATIONS	262
CREATING 3D MESHES WITH ENROUTE TOOLS	262
APPLYING MESHES TO A RELIEF.....	264
17. MODIFYING AND COMBINING RELIEFS	269
INTRODUCTION.....	269
SELECTING RELIEFS.....	269
CUTTING, COPYING AND PASTING RELIEFS.....	269
DELETING RELIEF OBJECTS	270
CLEARING RELIEFS.....	270
MOVING RELIEFS	270
ROTATING RELIEFS	271
SCALING RELIEFS.....	272
FIT RELIEF TO PLATE.....	273
SMOOTHING RELIEFS	273
INVERTING RELIEFS.....	274
CREATE A MESH SURFACE FROM A RELIEF	277
MERGING RELIEFS	278
COMBINE RELIEFS.....	280
EXTRACTING SLICES	280
18. USING BITMAPS	285
INTRODUCTION.....	285
APPLYING BITMAPS TO A RELIEF	285
USING A MASK	287
3D EFFECTS USING BITMAPS	288
19. TEXTURE TOOLS	291
INTRODUCTION.....	291
APPLYING TEXTURES TO A RELIEF.....	291
20. RELIEF EDIT TOOLS	297
EDIT RELIEF TOOLBAR.....	297
FADE AND SHRINK.....	298
BUILD PARAMETER.....	299
21. 3D TOOLPATHS.....	301

INTRODUCTION	301
3D TOOLPATH STRATEGIES	301
3D TOOLPATH STRATEGY PARAMETER.....	302
3D TOOLPATHS WITH A STEP ROUGH	311
ENGRAVE TOOLPATHS ON A RELIEF	318
CREATE RENDERED VIEW OF TOOLPATHS	322
22. AUTOMATIC TOOLPATHS	325
INTRODUCTION	325
DESIGN APPLICATION SUPPORT	326
OVERVIEW OF THE ATP PROCESS.....	326
ACTIVATING THE ATP DIALOG	327
THE ATP DIALOG	328
SELECTING THE ACTIVE DESIGN APPLICATION	329
SELECTING FILES TO PROCESS	330
LAYER MAPPING	332
ORDERING AND NESTING	338
ATP SETUP.....	344
EXAMPLE STEPS TO PROCESS YOUR FILES	347
KEYBOARD SHORTCUTS	351
INDEX	353

Software License Agreement

Carefully read the following terms and conditions sign and return as accepting these terms and conditions.

This is a legal agreement between you, the end user (either an individual or an entity), and SA International, Inc. If you do not agree with the following, you should promptly return the package. Use of this program indicates your acceptance of the terms and conditions stated below.

The enclosed computer program "Software" is licensed, not sold, to you by SA International, Inc., for use on a non-exclusive, non-transferable basis, only under the following terms, and SA International, Inc., reserves any rights not expressly granted to you. You may not disclose to any third party any confidential information concerning the Software or SA International, Inc. or use such confidential information to the detriment of SA International, Inc...

1. License.

This software is protected by the United States Copyright Law and International Treaty Provisions. Therefore, you must treat the Software just as you would any other copyrighted material, such as a book. This license allows you to:

- (a) Make one copy of the Software in machine readable form; provided that such a copy of the original may be used solely for backup purposes. As an express condition of this License, you must reproduce on each copy of the SA International, Inc., copyright notice and any other proprietary legends on the original copy supplied by SA International, Inc.
- (b) Transfer the Software and all rights under this License to another party together with a copy of this License and all written materials accompanying the Software provided you give SA International, Inc., written notice of the transfer and the other party reads and agrees to accept the terms and conditions of this License.
- (c) Use this Software on a single computer only, but may transfer it to another computer as long as it is used on only one computer at a time. "In Use" constitutes being loaded onto either temporary (i.e., RAM) or permanent memory (e.g., hard disk, CD-ROM or other storage device) of a computer.

2. Restrictions.

You may NOT distribute copies of the Software to others or electronically transfer the Software from one computer to another over a network. You may not de-compile, reverse engineer, disassemble or otherwise reduce the Software to a human perceivable form. You may not modify, adapt, transfer, rent, lease, loan, resell for profit, distribute, network or create derivative works based upon the software or any part thereof.

3. Termination.

This License is effective until terminated. This License will terminate immediately if you fail to comply with any of its provisions. Upon termination, you must return the Software, and all copies thereof, to SA International, Inc., and you may terminate this License at any time by doing so.

4. Export Law Assurances.

You agree that neither the Software nor any direct product thereof will be transferred or exported, directly or indirectly, into any country prohibited by the United States Export Administration Act or any international export laws and the restrictions and regulations thereunder, nor will it be used for any purposes prohibited by the Act or laws.

5. Warranty Disclaimer, Limitation of Remedies and Damages.

In no event will SA International, Inc., be liable for any damages, including infringement, lost data, lost profits, cost of cover or other special, incidental, consequential or indirect damages arising from the use of the program however caused and on any theory of liability. This limitation will apply even if SA International, Inc., or an authorized dealer or distributor has been advised of the possibility of such damage. SA International, Inc. MAKES NO WARRANTY, EXPRESS OR IMPLIED, WITH RESPECT TO THE SOFTWARE, AND DISCLAIMS, WITHOUT

LIMITATION, ANY IMPLIED WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. SA International, Inc. does not warrant any drivers for plotting, scanning or either devices. These drivers are provided for our customers as a service only, and were developed using information provided to us at the time by the equipment manufacturers.

SA International, Inc. is not responsible for any typographical errors in the software or in the documentation.

6. General.

If you are a U.S. Government end-user, this License of the Software conveys only "RESTRICTED RIGHTS," and its use, disclosure, and duplication are subject to Federal Acquisition Regulations, 52.227-7013(c) (1) (ii). This License will be construed under the laws of the State of Pennsylvania, except for that body of law dealing with conflicts of law, if obtained in the U.S., or the laws of jurisdiction where obtained, if obtained outside the U.S. If any provision of this License is held by a court of competent jurisdiction to be contrary to law, that provision of this License will remain in full force and effect.

© Copyright 2002 by SA International, Inc. All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted, in any form or by any means, electronic, mechanical, photocopy, recording or otherwise, without the prior written permission of the publisher. Printed in the United States of America. The information in this manual is subject to change without notice and does not represent a commitment on the part of SA International, Inc.

Acrobat® Reader Copyright© 1987-2002 Adobe Systems Incorporated. All rights reserved. Adobe and Acrobat are trademarks of Adobe Systems Incorporated which may be registered in certain jurisdictions. PostScript® software Copyright© 1984-1998 Adobe Systems Incorporated. All rights reserved.

Flexi, FlexiFAMILY, FlexiSIGN-Pro, FlexiSIGN Plus, FlexiEXPERT, FlexiSIGN, FlexiLETTER, FlexiDESIGNER, FlexiCUT, FlexiENGRAVE, PhotoPRINT Server, PhotoPRINT, PhotoPRINT SE, EnRoute-Pro, EnRoute Plus, EnRoute, EnRoute-Machine Shop, and/or other SA International products referenced herein are either trademarks or registered trademarks of SA International, Inc. Illustrator is a registered trademark of Adobe Systems Incorporated. FreeHand is a registered trademark of Macromedia Corporation. CorelDRAW! is a trademark of Corel Systems Corporation. AppleTalk, ImageWriter, LaserWriter, and Macintosh are registered trademarks of Apple Computer, Inc. Windows is a registered trademark of Microsoft Corporation. The names of actual companies and products mentioned herein may be the trademarks and/or registered trademarks of their respective owners. Adobe® is a trademark of Adobe Systems Incorporated or its subsidiaries and may be registered in certain jurisdictions. PostScript® is a trademark of Adobe Systems Incorporated or its subsidiaries and may be registered in certain jurisdictions.

SA International, Inc.

1. Welcome

Welcome to EnRoute

EnRoute is a design, toolpath generation, and output application that will allow you to create a wide range of finished products. With it you can create two-dimensional (2D) and three dimensional (3D) designs and then create all of the toolpaths necessary to produce the parts using your CNC machine.

EnRoute includes many productivity-enhancing features that simplify the design process. Templates can be saved at many points in the process so that your decisions regarding material types, tool choices, toolpath strategies, and output ordering may be saved for future use.

The goal of your design process is to produce finished parts efficiently and economically. EnRoute's wide range of toolpath creation tools, nesting, and output ordering options help you accomplish this goal.

This manual is intended to serve as a starting point for learning EnRoute. It is also available in the form of online Help in the software. Many other training options are available from SAI for those who would like additional assistance in taking advantage of EnRoute's capabilities.

In this chapter, you will learn about:

- What's new in EnRoute
- Recommended System Requirements
- Installing a hardware key
- Installing the Software
- EnRoute 4 Support Services

What's new in EnRoute

Version 4 of EnRoute includes many significant changes and improvements from previous versions. Virtually all areas of the software, from the design tools to the toolpath capabilities to the output capabilities have been enhanced. The following list of changes represents the most significant enhancements. Each of these is explained in more detail throughout this manual.

Drawing Tools

Text – The text tool allows the user to place text and then edit it to change parameters such as size, font, style and spacing. The user is able to select text to place on a curve or an arc.

Dimensions – EnRoute has several new dimensioning tools that allow the user to dimension lengths, angles radii and diameters.

Snapping – New snapping capabilities have been added and are available in more tools.

Drawing Tools – The Rectangle, Circle, Ellipse Arc and Polygon tools have been recreated to make them more capable and easy to use.

Poly-Arc Tool – This new drawing tool will make it much easier to draw a polyline comprised of lines and arcs. It allows the user to create each segment either graphically or precisely, and switch

between arcs and lines easily.

Node Editing - The node editing tool will allow zooming and panning within the tool.

Move, Rotate, Scale – New tools have been created that allow for better precision.

Measure – A new measuring tool allows the user to measure distances, angles, and areas.

Partial Offset – This offset tool has been improved to allow the user to select sections of a contour to offset, as well as open contours.

Offset – The contour offset tool now provides a preview of the offsets before they are accepted.

Fillet – The fillet tool offers more options, provides a preview of the fillet, and allows either individual fillets or for an entire contour to be filleted.

Mirror – The new mirror tool provides more options, including the ability to define the axis around which to create the mirror, and the option to create a copy with the mirror. Also, a preview of the mirrored object is created.

Nesting Report - EnRoute provides the option to have a nesting report generated that provides information about the efficiency of the nest.

Dynamic Nesting – EnRoute has a new nesting tool that allows the user to drag objects onto the plate and have them automatically rotate to fit within the constraints of other objects and the edge of the plate. Several parameters allow you to adjust offsets and rotations steps.

Reliefs

Relief Editing – EnRoute has a suite of new relief editing tools that allow the user to interactively edit a selected relief. These powerful tools provide a new creative option for relief creation and editing.

Textures – EnRoute has a new parametric texturing engine that provides a new and unique way to add textures to surfaces without the need to use bitmaps. The new texturing tools create textures that go out in all directions, automatically match the resolution of the relief, and are parametrically defined which allows the user to create unique effects.

Baroque Chamfer Tool – This is a unique tool that allows the user to simulate the effect of a traditional hand-carved surface. It automatically determines convex and concave sections of the artwork and then applies different profiles to these sections, with smooth transitions between them.

Relief Rendering – EnRoute has improved the rendering of reliefs in all of the views. Also available are lighting options so that you can change how the relief is lit.

Toolpaths

Spiral Fill – EnRoute has a new spiral fill option that will be useful in certain applications.

Bridges – EnRoute has implemented several new types of bridges that improve the smoothness of movement of the tool as it creates the bridge.

Bridge Editing – the Bridge editing functions have been improved.

Automatic Toolpath (ATP)

DXF Part Quantities – You will be able to specify the quantities of parts you want to process using DXF files that are not associated with a specific design application.

Cut List Sorting – You are able to sort parts in the cut list by clicking on the column headers.

Reporting – EnRoute has worked to improve the organization and presentation of the printed report that is generated showing the nested sheets.

Recommended System Requirements

Before you begin installing the software, read the hardware requirements below. For optimal performance we suggest that your system meet the recommended requirements. As with all computer software, systems with faster processors, more RAM, and greater amounts of storage space allow you to work with larger files and keep your processing time to a minimum.

	Windows
Processor	Pentium 4 or better
RAM	1 GB
Install Space	300 MB
Working Disk Space	4 GB
Operating System	Windows 98/ME/NT4.0/2000/XP/Vista
Video	1024 x 768 resolution with 24-bit color
Other	DVD drive
	Free USB port for hardware key

Hardware Key

The hardware key is provided with each software license. It is required, along with an appropriate password, in order to run the software. The hardware key serves two purposes. First, it protects the software from being unlawfully copied and distributed. Second, it provides a simple way for you to transport your software license between computers. It is acceptable to install EnRoute on more than one computer and then use the hardware key to move between computers.

EnRoute 4.1 requires the use of a SafeNet USB key. If you are using an earlier version of EnRoute that utilized a different type of hardware key it will be necessary to exchange your key for a SafeNet Sentinel key. If you are already using a SafeNet key then this is not an issue.



SafeNet Sentinel USB Hardware Key

It is necessary to install a hardware key driver in order for the key to function properly. This can be accomplished as part of the software installation process. It is only necessary to install the hardware key driver one time on a computer.

Installing the Software

If you received an installation CD, then place the CD in the CD drive on your computer. The installation should start after several seconds. If the installation does not start automatically, do the following:

1. In Windows, from the Start menu choose **Run**.
2. Type D:\setup.exe and click the **OK** button. (The letter D represents the CD drive on your computer. Use the appropriate letter for your CD drive.)
3. The installation process should start.

It is also possible that you may have received an EnRoute installer via email or by downloading it from the EnRoute web site. In this case you can start the installation process by double-clicking on the installer file.

The EnRoute installer will lead you through several steps to install the software in the desired location on your computer. You will also have the opportunity to install the appropriate password for your version of EnRoute during the installation process.

Following are the installer steps along with an explanation of the appropriate action.

- | | |
|--|--|
| 1. Welcome to the EnRoute Setup Wizard | Click Next to continue. |
| 2. License Agreement | This provides the agreement between you and SA International, Inc. for the license of EnRoute. You are encouraged to read this agreement in full. If you agree, click the I Agree button to continue with the installation. |
| 3. Choose Install Location | The default location is c:\EnRoute4\. EnRoute is a self-contained application so all of the files required to run the software are included in the install folder. You can choose another location or another name for the EnRoute folder. After choosing the desired install folder click Next . |
| 4. Choose Start Menu Folder | The default folder name is EnRoute4_1. You may choose to call it something else if you wish. After defining the start menu folder name, click Next . |
| 5. Key Driver Installer Option | You may choose to install the Hardware Key Driver in this step. If you are already using a SafeNet key like the one pictured in |

the previous section, you likely do not need to install the key driver. If you are installing EnRoute for the first time, or if you are using a SafeNet key for the first time, it will be necessary to install the key driver. Installing the driver again even if it is already installed will not cause a problem. Choose the desired option and click **Next**.

6. Password Installer Option You may choose to install the EnRoute password in this step. Password installation is required before you are able to run EnRoute, but you may also install the password after EnRoute has finished installing. Choose the desired option and click **Install**.
7. Installation of the software will start
8. Do you prefer to work in millimeters? You will be prompted with this question. If you do most of your work in millimeters, click **Yes**. Some of the initial default values in EnRoute will be defined appropriately based on your choice.
9. Hardware Key Installer If you chose to install the key driver, that process will be executed now. This requires a few installation dialogs. Follow the instructions to install the driver.
10. Password Installer If you chose to install the password, that process will be started now. Enter your 32-digit password carefully in the spaces provided. After you have completely entered the password, the OK button will be enabled. Click OK. A dialog
11. Installation Complete After all of the files have been installed, you will be notified that the installation is complete. Click **Next** and then click on **Finish** in the dialog that will exit the installation wizard.

EnRoute 4 Support Services

On-Line Support Services

SA International recognizes the pressures you must endure with you business, computers and new software. There are times when you need help, which we understand. SA International is available to help you with your technical question, as well as your "How To" problems with our software. A variety of help methods are available to fit your budget, as well as your urgency to offer true value and peace of mind.

Support in North America

New and upgraded products qualify for 90 Days Free Support from the date of purchase. During these first 90 days, unlimited free technical support is available in North America by calling our Technical Support Team Line: **800.229.9044**

Technical Support Hours: Monday – Friday 8:00 AM – 5:00PM MST

E-Support: Esupport@SAintl.biz

Questions regarding current version products can be submitted to our e-Support at no charge at any time. Replies are generally made within one business day.

Fax Support:

Questions regarding current version products can be submitted to our Technical Support Fax line at anytime. Replies are generally made within one business day.

Contact

Fax Support Line: 801.401.7840

Corporate Office

1490 N. 2200 West, Suite 120

Salt Lake City, UT 84116 USA

Tel: 800.229.9066

Tel: 800.229.9068

Fax: 801.401.7234

European Office

Minevastraat 14 Bis

B-1930 Zaventem, Belgium

Tel: +32.2.725.5295

Fax: +32.2.725.2809

Getting Started with EnRoute Instructional DVD

This interactive DVD featuring Tom Penner contains more than 4 hours of over 70 training videos, covering a wide array of topics that include:

The Basics, Advanced Drawing Tools, 2D Toolpaths, 3D Engraving Toolpaths, Introduction to NBM, Introduction to 3D Surfaces, and much more— all designed to get you up and running with EnRoute Software.

This DVD is available for purchase through the web site: <http://www.saintl.biz> Go to the training section to place your order.

2. Getting Started

Workflow

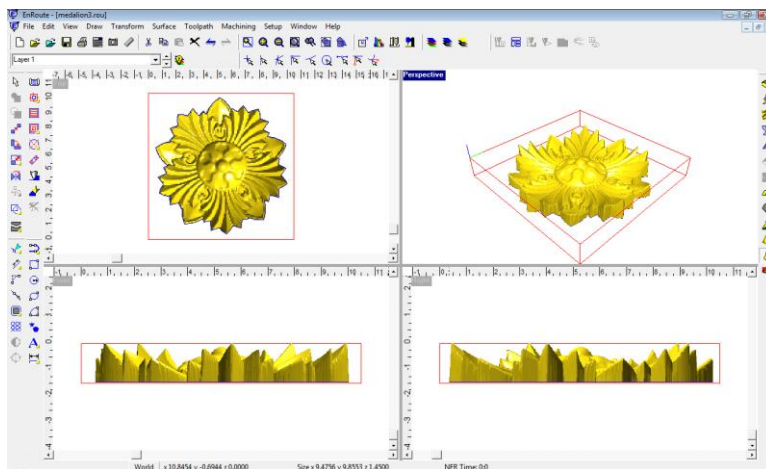


The graphic above illustrates the basic concept for completing a project in EnRoute. It may seem overly simplistic, but it does have a valuable practical use. When you are working in EnRoute, you are at all times within one of these steps of the process, and it is important to always understand where you are in the process. In the Design phase you are typically creating 2D contours for your design. You might also be creating a relief surface. Everything you do to define the geometry and surfaces of your design is part of the design process.

After completing the design, you are ready to create the toolpaths for cutting out the design. In this stage, it is important to understand such things as the material you plan to use, the tools you have available, and the quality of the finish required on the cut edges and surfaces of the parts. Output to the machine requires you to select the correct output driver, and to make sure that the configuration in EnRoute matches your specific machine.

Basic Elements of EnRoute

The main display area shows a view of the workspace. It can display a Top, Right, Front or Perspective view of the workspace, or all four views at the same time. Although you will likely do the majority of your work in the top view, it is often very convenient to be able to view contours and toolpaths from other views in order to see how they relate to each other.



Working with Views

To maximize a view, double-click on the name of the view that appears in its upper left corner. Double-clicking on the name a second time returns you to the four-view layout.

You can also toggle between the top view and the perspective view using the **F12** key.

Menus

All tools and options are available by selecting them from the menus at the top of the screen. **File Edit, View**, etc. Click on the Menu name to open the menu.

Toolbars

In addition to locating the tools through the menus you can also find the tools readily available on the toolbars. Some of the toolbars contain flyout Toolbars. The flyout Toolbar is a group of tools that share the same space on a toolbar, the entire group of tools “fly out” when you press and hold the first tool. A small yellow triangle in the lower right corner identifies the first tool in a fly-out toolbar.

Hiding and Displaying Toolbars

Menu: Setup / Toolbars

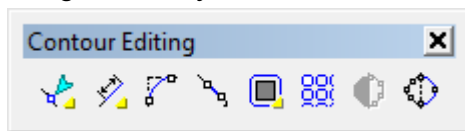
The user has the option to turn off some or all of the toolbars associated with this program. If there is a check mark in front of the toolbar’s name the toolbar is turned ON.

Resetting Toolbars

Menu: Setup / Toolbars / Reset to Defaults

Select **Reset to Defaults** if you would like to set the toolbars back to their original position.

Using Toolbar Flyout Menus

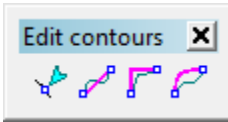


Many toolbar icons have a small yellow triangle in their lower right corner indicating that a flyout menu for the tool exists.

A quick click on the toolbar icons will activate that particular tool. Click and hold for a short time in order to access the flyout menu containing several more tools.



If you click on the edge of a flyout menu and drag it, the menu will “tear off” and form a floating menu.



The original flyout menu will still be there: the floating menu is a duplicate.

Rulers

Menu: View / View Ruler

Rulers appear along the top and left side of the main screen to help you measure and align contours. Rulers can be activated for the top, right and front views from the View menu. The rulers can be very helpful in keeping track of where you are in a design as you move around in the design during the design process.

The rulers are also a handy way to bring guidelines into the work area, by just clicking on a ruler and dragging the guideline into the design.

Displaying and Hiding Rulers

Menu: View / View Ruler

To show or hide the rulers, from the **View** menu, point to **View Ruler** and click **Top, Right or Front view**.

Changing Units

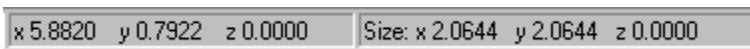
Menu: Setup / Preferences / Units tab

To change the units used for all the drawing elements in your design, from the View menu, click on **Preferences** and choose the **Units** tab. Select the unit of measure desired.

Status Line

The status line along the bottom of the display provides a number of pieces of information. On the left side of the status bar the current coordinates of the cursor are constantly updated. If objects are selected, their size is displayed to the right of the coordinates.

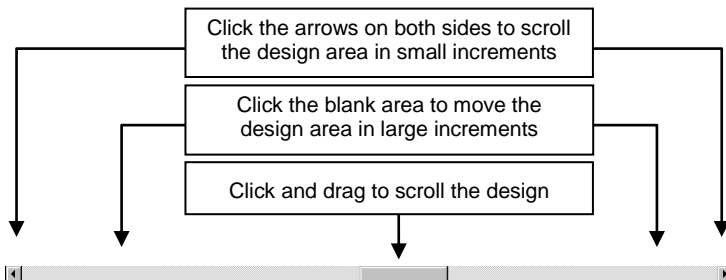
During operations that require more than a moment or two of processing, a status bar is activated to provide feedback of the progress of the task.



Using Scroll Bars

Menu: View / View Scroll Bars

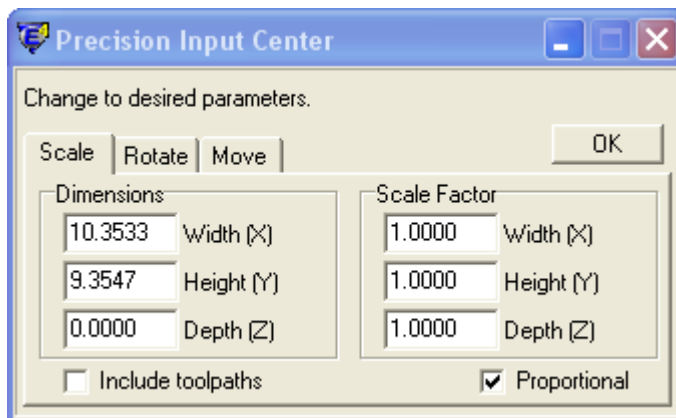
Scroll bars are horizontal and vertical bars located on the bottom and left side of your design area. Use them to scroll the design area.



To toggle scroll bars on and off, from the **View** Menu, select the **View scroll bars** option.

Precision Input Center

The Precision Input Center is utilized as an interface for some of EnRoute's activities. It can be activated at any time by pressing **F2**. Activating the Precision Input Center by using **F2** provides a simple method to move, scale or rotate selected objects. Other, more detailed, methods for moving, scaling, and rotating selected objects are also available, and are discussed in other sections of this manual.



For example, if you want to move the selection to a specific location, press **F2** and select the **Move** tab. Then enter the coordinates for the new location of the selection.



Note also the Reference Grid control, which allows you to choose which corner, side or center point on the selection, will be lined up over the specified point when the selection is moved.

Using Snaps

Menu: Setup / Snaps

Snaps allow you to select specific points during geometry construction and editing activities. Within most of the drawing commands, if you have snaps activated then as you move the cursor around on the screen the cursor will snap to other drawing element points that meet the criteria of the snap. For example, if you have Snap to Endpoint on and then move the cursor close to a contour endpoint the cursor will snap to the point. Snapping makes it much simpler to precisely create and edit contours.



	Icon	Description	Cursor
Snap to grid		Snaps to the nearest grid point. The grid spacing is configured on the Grid page in Preferences.	
Snap to guideline		Snaps to a nearby guideline when the cursor gets close. Also when dragging a contour, the perimeter of the contour snaps to nearby guidelines.	
Snap to intersection		Snaps to the intersection of two line segments.	
Snap to endpoint		Snaps to the endpoints of arc, curve and sine segments.	
Snap to nearby contour		Snaps to the nearest point on a contour when the cursor gets close to it.	
Snap to center of arc		Snaps to the center of an arc segment when the cursor is placed near the arc.	
Snap to midpoint		Snaps to the midpoint of a segment when the cursor gets close to it.	
Snap to perpendicular point		When constructing line segments, after defining the first point in the segment, the second endpoint will snap to a point on a contour so that the line segment will be perpendicular to the contour.	

Snap to tangent point



When constructing line segments, the second endpoint will snap to a point on an arc so that the new line segment will be tangent to the arc.



The snap distance value is set on the General Tab of the Preferences Dialog. See “Setting Preferences” for details.

Using Guidelines

Menu: View / Edit Guidelines

Guidelines are design aids that display as a dotted line in your design view. If **Snap to Guidelines** is enabled, contours moved near a guideline will automatically be positioned along the guideline.

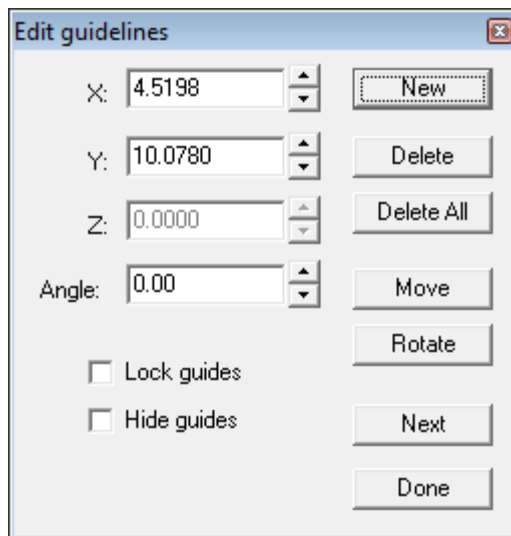
- Clicking on the **Next** button will allow you to move through the guidelines that have been placed. The guideline will be highlighted in blue to show which guideline has been selected.

Edit Guidelines Dialog

Menu: View / Edit Guidelines

Use the Edit Guidelines Dialog to Create, Delete, Move, Rotate, Lock or Hide a guideline.

To open the Guideline Dialog: Select **Edit Guideline** from the **View** Menu.



Creating a Guideline

Menu: View / Edit Guidelines

The simplest way to place a guideline is to 'drag' it from the horizontal or vertical rulers on the screen. Place the guideline by left-clicking on the ruler and then dragging the guideline out into the design.

You can also create a new guideline using the Edit Guideline dialog:

1. Activate the view in which the guideline is to be placed by clicking in that view.
2. Enter in the coordinates and angle of the new guideline.
3. Click on the **New** button.
4. Click on **Done** to exit the dialog.

Deleting a Guideline

Menu: View / Edit Guidelines

1. Activate the Guidelines dialog.
2. Click on the **Next** button until the guideline is highlighted in blue.
3. Click on the **Delete** button.



It is also possible to right-click on a guideline and then select **Delete guideline** from the shortcut menu. You cannot delete a guideline by dragging it out of a view.

Moving a Guideline

Menu: View / Edit Guidelines

1. Activate the Guidelines dialog.
2. Click on the **Next** button until the guideline is highlighted in blue.
3. Enter the coordinates for the new location.
4. Click on the **Move** button.



Guidelines may also be moved interactively by simply selecting the guideline with the mouse...left click, hold and drag to the desired location. If the lock guides option has been enabled this interactive method does not work.

Rotating a Guideline

Menu: View / Edit Guidelines

1. Activate the Guidelines dialog.
1. Click on the **Next** button until the guideline is highlighted in blue.
2. Enter the angle (in degrees) you would like the guideline to be rotated to in the **Angle** field.
3. Click **Rotate**.
4. Click **Done**.

Locking Guidelines

Menu: View / Edit Guidelines

The **Lock Guidelines** feature prevents guidelines from being moved or edited by accident. Guidelines may be added while the guidelines are locked, but may not be deleted, moved or edited.

1. Activate the Guidelines dialog.
2. Check the **Lock Guides** box.
3. Click **Done**.

Hiding Guidelines

Menu: View / Edit Guidelines

The Hide Guidelines feature allows you to hide the guidelines without actually deleting them. While this feature is enabled, no guidelines will be displayed, the Snap to Guidelines feature is disabled.

1. Activate the Guidelines dialog.
2. Check the **Hide Guides** box.
3. Click **Done**.

Changing the View

The commands available in the **View** menu and the **Zoom** toolbar allow you to zoom in and out to get a closer look at part of your design or see the whole thing. The zoom commands do not change the size of any part of your design; they simply change your view of the information.

Zooming with the Mouse

To zoom in and out using the mouse, hold down the **CTRL** key and drag with the right mouse button. Dragging the mouse up or left zooms out, down or right zooms in.








Zooming using a Wheel Mouse

Zooming with a wheel mouse is a particularly handy way to get around a drawing. Move the cursor to where you want to work, and move the wheel backward to zoom out or forward to zoom in. Zooming is centered on the cursor location which makes it very easy to zoom into a particular location.

Zoom Commands

Menu: View

Toolbar: Zoom

Function	Icon	Description
Zoom Window		<p>The Zoom Window command allows you to select an area to magnify.</p> <ol style="list-style-type: none">Click with the mouse and drag a box around the area you want to magnify.Release the left mouse button and the software will zoom in on the selected area.
Zoom In (Ctrl + I)		<p>Zoom in towards the center of the window.</p>
Zoom Out (Ctrl + K)		<p>Zoom out and enlarge the view of the design.</p>
Zoom to Plate (Ctrl + P)		<p>Display the entire plate.</p>
Zoom Previous		<p>Return to the previous zoom setting.</p>
Zoom to Extents of Selection		<p>Zoom in on the currently selected contour or contours.</p>
Zoom to Extents of all Objects		<p>Zoom out to display all contours, including the plate if one is designed.</p>

Redraw

Menu: View / Redraw

The Redraw Tool is used to redraw or refresh the screen and clear any remnants left over from manipulating drawing elements, from the **View** menu, select **Redraw** or press **Ctrl + R**.

View Setup


Menu: View / Setup

Toolbar: Setup / View Tab

The View Setup tab of the Preferences dialog determines how information is displayed on screen. To access the View Setup tab, from the **View** menu, select **View Setup**.

Using the Undo and Redo Features


Menu: Edit / Undo

To undo the last operation, from the **Edit** menu, select **Undo [...]** or type **CTRL+Z**. . The name of the last operation is displayed after the Undo command.

If you decide that you didn't want to undo an action, from the **Edit** menu, select **Redo [...]** or type

CTRL+SHIFT+Z.



 If you have more than one design open, the Undo and Redo commands only affect changes made to the active design.

Working with Layers

Menu: Setup

Toolbar: Settings

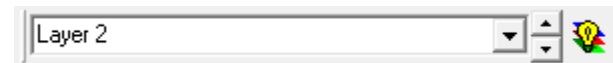
EnRoute offers the ability to organize your designs into Layers. Layers provide a way to separate design information and toolpaths into groups. Individual layers may be displayed or hidden at any time. In addition, contours contained in different layers can be displayed in different colors to help distinguish different parts of the design.


Toggle Layers



Toolbar: Settings

To activate the layers toolbar click on the layers toggle icon. This will open the layers toolbar. The layers can be easily managed using the arrows to access the different layers.



By clicking on the **Activate all layers** icon  you can easily switch between the current layer displayed and displaying all of the layers.

Layers Dialog

Menu: Setup

Toolbar: Settings

Each layer has several options for display and use. These options include: **On**, **Off**, **Lock** and **Move Lock**. A check in the box indicates that this function is turned on for this layer.

If the Layer is: **The Objects on the layer:**

On All objects on that layer are displayed and can be selected.

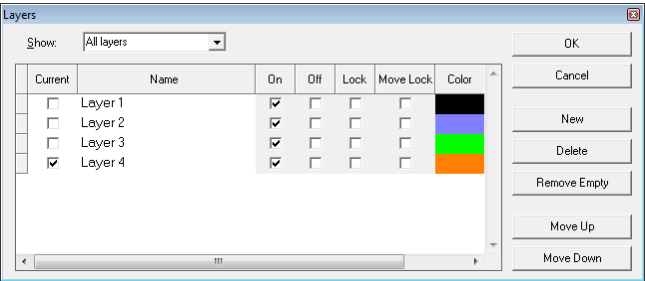
Off None of the objects on the layer are displayed, and none of the objects on that layer can be selected.


Lock

All of the objects on the layers are displayed but cannot be selected. You can still snap to the objects using the snap tools.
The current layer cannot be locked.

Lock Move

All of the objects on the layers are displayed, they can be selected but they cannot be moved.



To access the **Layers** dialog, click on the **Layers** icon 

To:

Do The Following:

Select the Current Layer

Check the box in the **Current** column for that layer.
The current layer is the layer in which any contours will be created.

Add a New Layer

Click **New**.
The layer is added to the bottom of the list, and is automatically selected as the current layer.

Delete Layers

Select the layer and click **Delete**.
If there are contours contained in that layer, you will be asked to confirm your action. If you delete the layer, its contents will also be deleted.

Remove Empty

Click on the Remove Empty button to remove all layers that are empty. This is useful in helping to clean up a drawing by removing empty layers.

Rename a Layer

Edit the text in the **Name** column for that layer.
Highlight and type the new name.

Change the Color of a Layer

Click on the color displayed in the **Color** column for that layer. A standard Windows color selector window will open.
Select the new color and click **OK**.

Moving Layers

To move a layer, select the layer and click on **Move Up** or **Move Down**. You can also move a layer by dragging the row header at the left edge of the list.

Hide or Display Layers

In the On and Off column, check **On** to display a layer, **Off** to hide it. The current layer cannot be hidden.

Lock a Layer	Check the box in the Lock column to lock a layer. Contours contained in a locked layer cannot be edited or deleted.
Unlock a Layer	Clear the box in the Lock column to unlock the layer. The current layer cannot be locked.
Save Changes and Exiting	Click OK to save all changes and close the dialog.
Cancel Changes	Click Cancel to close the dialog without saving any changes.


Changing a Contour's Layer

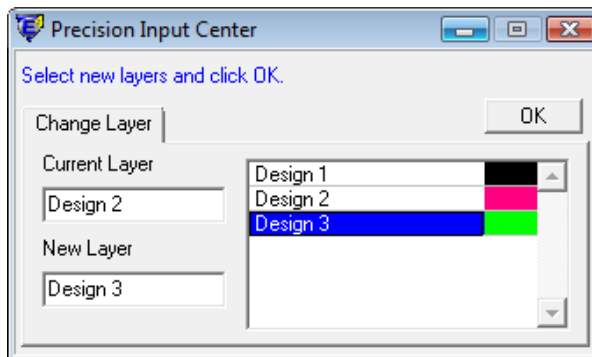
Menu: Transform / Change Layer

Toolbar: Settings

When a contour is created, it is placed on the current layer.

To change a contour's layer:

1. Select the contour(s).
2. Activate the **Change Layer**  dialog either from the menu or from the toolbar button.



3. Click on the layer you want to move the contours into.
4. Click **OK**.

Using Help

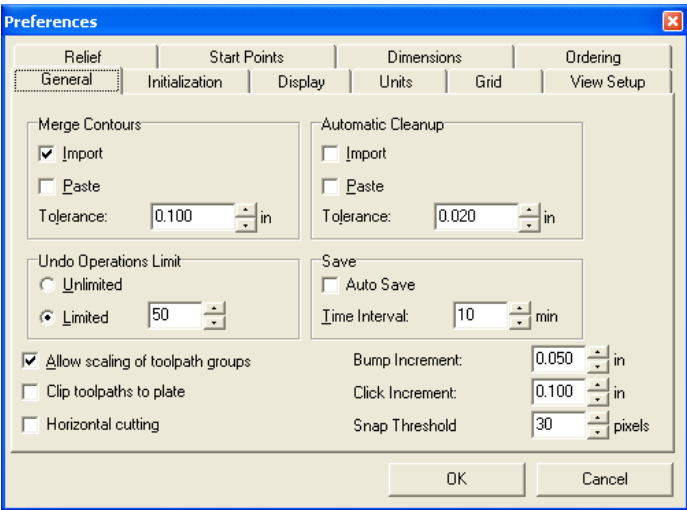
Menu: Help

To display the EnRoute Help system, from the **Help** menu, select **Help Contents**. You can also press **F1**.

Setting Preferences

Menu: Setup / Preferences

EnRoute allows for several categories of options to be individually chosen by the user. Access the preference settings in the Setup Menu. Enter your preferred settings. Click **OK** to save changes. Click **Cancel** to close the dialog without saving changes.



The available settings are:

General Tab

Merge Contours

Designs which were created and exported from other drawing programs are not always created as closed contours. This function provides for and automatically merges contours together.

Import Check this box to merge contours when they are imported.

Paste Check this box to merge contours when they are pasted from the clipboard.

Tolerance The maximum separation between contour endpoints that will still be merged.


Automatic Cleanup

These options specify whether the program should automatically delete short segments when importing or copying contours into the program.

Import Check this box to delete segments when contours are imported.

Paste Check this box to delete segments when contours are pasted from the clipboard.

Tolerance The maximum length of segments that will be deleted.

Undo Operations Limit	<p>Sets the number of operations that are saved so that the Undo command can undo them.</p> <p>Unlimited All operations since the last save can be undone.</p> <p>Limited The Undo command will be able to undo only the specified number of operations.</p>
Save	<p>The Auto Save function automatically saves the current design at the specified time interval.</p> <p>Auto Save Check to enable the AutoSave function.</p> <p>Time Interval The time interval at which the design is automatically saved.</p>
Allow scaling of toolpath groups	<p>Check to allow groups of toolpaths to be resized as a group. If cleared, toolpath groups cannot be resized. Checked by default.</p>
Clip toolpaths to plate	<p>If checked, output toolpaths will not extend outside the boundary of the plate.</p>
Horizontal Cutting	<p>If this option is available, it allows you to control whether you want to allow toolpaths to be rotated out of the horizontal x-y plane. This is only appropriate if your machine has the ability to cut toolpaths that are not in the x-y plane – most likely using an aggregate head. If this option is available in your version of EnRoute, it should most likely remain unchecked unless you specifically need to rotate toolpath groups.</p>
Bump Increment	<p>Set the amount that a selection is moved when an arrow key is pressed.</p> <p> When moving a contour using the arrow keys to bump it, it is normally bumped by the increment specified here. However by holding down the shift key, the bump becomes a portion of the display width, so it gets smaller as you Zoom In.</p>
Click Increment	<p>Sets the amount that values in a field are changed by when the “spinner” to the right of the field is clicked.</p>
Snap Threshold	<p>The distance in pixels at which the snap function is triggered, snapping a contour to a grid point or other positioning guide.</p>

Initialization Tab

Maximize Application	<p>Check this box to automatically maximize the software on startup.</p>
Maximize Document	<p>Enable this function to automatically maximize new documents when they are opened.</p>
Display 4 Views	<p>Check to display designs in the 4-view mode by default.</p>
Prompt for Plate	<p>Check to open the Define Plate dialog automatically when a new design.</p>

Small Part Size Threshold	This parameter is used by EnRoute for selecting 'small parts' when the option to output small parts first is selected in the output dialog.
Solutions Path	This defines the path of any installed 3 rd party applications that provide additional functionality to EnRoute.

Display Tab

The Display tab allows the user to define which colors are associated with the different types of drawing elements.

To change a color:


1. Double-click on the color you want to change.
2. Select the color desired and click **OK**.
3. Click **OK** to exit and save any changes.

Show Layer Colors	Check this box to show the colors assigned to the individual layers.
Update Buttons	This should typically be checked. It allows EnRoute to update buttons so that buttons are only enabled when they are appropriate based on the current status of the selected objects. If it is not checked, then buttons will be left enabled even though they might not be functional.
Enable OpenGL	This parameter allows EnRoute to utilize the best drawing and rendering capabilities of the graphics card in your computer. It should only be turned off if there is a compatibility issue with the OpenGL implementation in your computer and EnRoute.

Units Tab

The Units tab allows you to select the units that will be used to measure **Length**, **Time** and **Speed** in the software.

If the units are changed, the measurements of any existing contours will be converted to the new units so that they remain the same size.

-  If the units are changed while a design containing toolpaths is open, the measurements of the toolpaths may not be converted correctly. For this reason, you should save and close the design, then change the units and reopen the design. This ensures the toolpaths will be converted properly.

Grid Tab

The Grid tab allows the user to determine the view of the grid shown on the document page. Using the grid is very helpful in determining the placement of contours. There are two levels of the grid that may be enabled and adjusted to the layout of a particular project.

Show Grid	A checkmark in this box indicates the grid is turned on.
Interval of the grid marks	The distance between the grid marks.

Style of grid marks	Select None , Cross , Tick , or Line .
Size	Size of the grid marks in pixels.
Color	Color of the grid marks.
Show grid in perspective	Check this box to display the grid in the Perspective view.

View Setup Tab

The View Setup tab determines how information is displayed on screen.

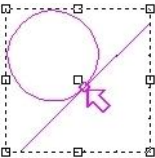
Standard Items

Plate Check this box to display the plate, if any.

Contours Check this box to display the contours.

- Since contours are the main type of drawing element, this is typically left on. Sometimes it can be convenient to turn contour display off to review only the toolpaths.

Contour Loops



Check this box to display contour loop indicators.

While converting a contour to a toolpath, the software will try to detect loops in the contours. If loops are detected, and you choose not to proceed (in which case the loops are automatically removed), the points at which the contour crosses itself will be indicated with contour loop indicators.

Open Contour Direction Check to display direction arrows on open contours.

Popup menu on Right Click Check to enable a right click with the mouse to bring up a popup menu for the tool you are using.

Toolpath Items

Toolpaths Check this box to display the toolpaths.

Direction Check this box to display direction arrows on toolpaths.

Entry/Exit Check this box to display the Entry/Exit positions for each toolpath.


Bridges Check this box to display the location of toolpath bridges.

Start Point Check this box to display a small circle with an X in the middle at the start point of each toolpath. If you want to see Lead In/Lead Out toolpaths make sure the Start Point is turned on also.

Toolpath Width The Width option determines how toolpaths are displayed on screen. There are two options:

Lines The toolpaths are displayed as lines with directional arrows (Default).

By Tool The toolpaths are displayed using the diameter of the tool used to create the toolpath.

 This option will give you a better idea of what the final piece will look like after it has been machined. The Simulate 2D command located in the Machining Menu will show the same thing as the By Tool option, in addition to showing the cutting sequence.

Depth	This option controls how the depth of the toolpaths affects the way they are displayed.
All Depths	All toolpaths at all depths are displayed (Default).
Surface	The toolpaths at the surface is displayed.
Final	The last (bottom) toolpath is displayed.
[Specific depth]	One of the specific depths assigned to the passes is selected for display.
Tools	This option selects which toolpaths will be displayed based on the type of tool associated with the toolpath.
All Tools	Toolpaths for all tool types are displayed (Default).
[Specific Type]	Only the toolpaths which use the selected type of tool are displayed.

Relief Tab

Toolpath Tolerance	This parameter defines how closely 3D toolpaths follow their relief. This value might vary depending on the size and resolution of the reliefs. The smaller the tolerance the more closely the toolpaths will follow the relief, but this will also create more segments in the toolpaths and increase the size of the output files.
---------------------------	--

Start Points

This tab allows you to define a preference for where start points are automatically located on Routing Offset toolpaths.

- Long Edge and Edge Midpoint**

This will locate the start point on the longest edge of the contour. If Edge Midpoint is checked, then the start point will be moved from the start of the long edge to its midpoint.
- Direction**

This option allows you to define a direction that EnRoute will then use to locate the start point. The direction is entered in the edit box to the right in degrees.
- Magnetic**


This option allows you to enter a specific point, and then EnRoute will create the start point for each object as close to that point as it can. This allows you to automatically pull start points toward an area of the material.
- Longest Segment**


This option picks the longest segment within the contour and locates the start point either at the start of the midpoint of the segment depending on whether the Segment Midpoint option is checked.


Dimensions Tab


The Dimensions Tab sets up the display parameters of the dimension tools. All dimensions in the drawing will be displayed using these parameters.


- Text


Select the font for the dimension text.
- 


This parameter sets the height of the text in the dimensions tool.
- 


This parameter sets the distance that the extension lines start to draw from the start and end points of the dimension.
- 


This parameter determines the height of the arrow.
- 

This parameter determines the width of the arrow.
- 

This parameter sets the number of decimal positions for the measurement of a line segment.
- 

This parameter sets the number of decimal positions for the measurement of an angle.
- 

This sets the position of the text above the arrow line.
- 

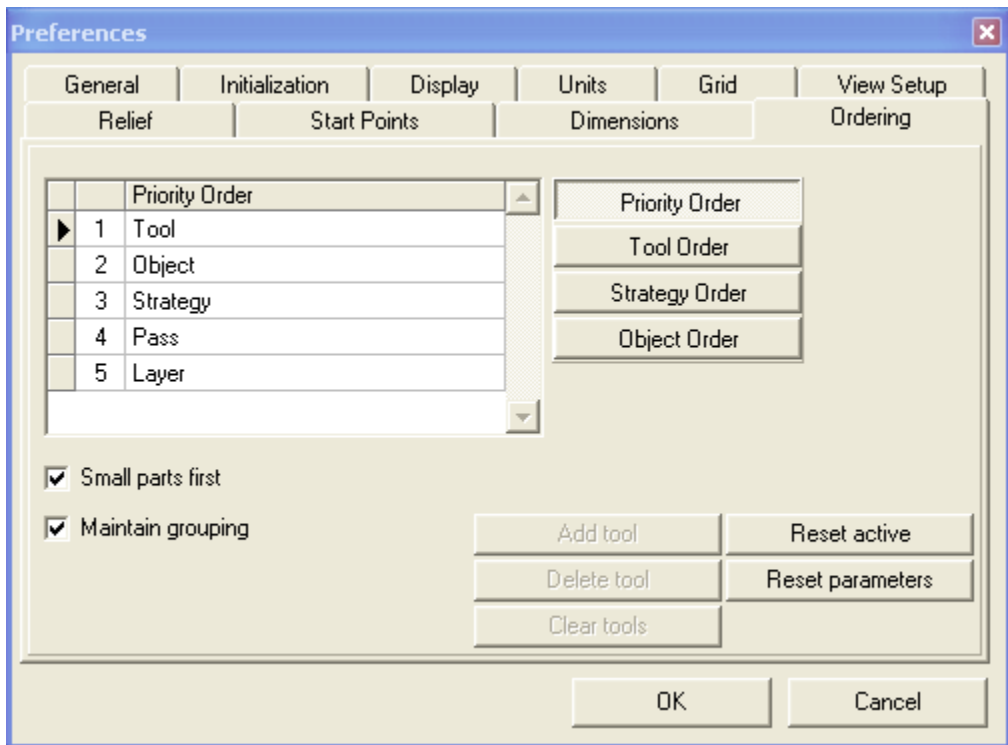
This sets the position of the text in the center of the arrow line.
- 

This sets the position of the text under the arrow line.

Ordering Tab

The Ordering tab allows you to define default ordering preferences so that when toolpaths are automatically ordered by EnRoute, the order will follow your preferences. The layout of this dialog is essentially the same as the ordering dialog that is presented in other tools such as Output, 2D Simulate, Ortho Simulate, and the Rendering tool. And, they are all related.

It is possible from each of those other ordering tools to reset the ordering to match these preferences, and it is also possible to send the current ordering settings in a file to the ordering preferences. For example, maybe you have defined your preferences so that Routing Offset strategies come before Drill strategies. And then you find that when you prepare your jobs for output you always change the Strategy Order so that the routing offsets go last. From within the output dialog you can easily make those updated choices become your ordering preferences. Then the next time you create a design that includes routing offsets and drills, the default ordering settings will reflect the updated preferences that have the routing offset after the drills.



The following table provides a brief description of each of the parameters in the Ordering Preferences dialog.

Priority Order	Set the priority order for the order in which toolpaths sorted. In the priority settings in the graphic above, the toolpaths will be sorted based on the tool order as the top priority, followed by the Object Order, Strategy Order, etc... A more detailed explanation of the ordering philosophy in EnRoute is located in Chapter 11 of this manual.
Tool Order	You can choose tools from EnRoute's tool library to add to the Tool Order list. You might typically add tools that you use often, and then click and drag them into the preferred order.
Strategy Order	This list includes all of the strategy types available in EnRoute. By default they are in a reasonable order – drills are first in the list and routing offsets are last. You can click and drag these strategy types into your preferred order.
Object Order	You can choose the default object ordering method, including shortest, rows, columns, inside out, and outside in.
Small Parts First	If this is checked, objects with a surface area less than the small part size threshold defined in the Initialization tab will be put at the top of the object order list regardless of the object ordering method.
Maintain Grouping	If objects are grouped together, this checkbox allows them to be treated as one object for object ordering.
Add Tool	This button allows you to add a tool to the Tool Order list.
Delete Tool	This button removes the current tool from the Tool Order list.
Clear Tools	This button clears all the tools from the Tool Order list.
Reset Active	This resets the currently active parameters (Priority, Tool, Strategy or Object) to their default values.
Reset Parameters	This resets all the ordering parameters to their default values.

Exiting the Software

Menu: File

From the **File** menu, select the **Exit** command to close all layouts and exit the program.

3. Toolbars and Command Icons

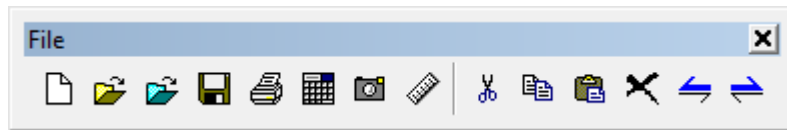
This section provides a summary of the major functions of EnRoute. As you will recognize, most of the commonly used tools are readily available by clicking the screen buttons available in the toolbars. The icons presented and described in this section are organized based on the toolbars in which they reside.










The toolbars can easily be moved to the top, bottom, right or left side of the display. This allows you to customize the interface to fit your preferences. The changes you make will be saved when you exit so that the layout will come back just as you left it the next time you start the program.

Until you become familiar with each of the icons, you can view a short “hint” about each icon by placing the cursor over the button and leaving it there for a second or two.

In addition to the screen icons, you can also reach all of the EnRoute functions by using the pull down menus.

File Toolbar



-
- | | |
|---|--|
|  | New (Ctrl+N) - Use the New command to start a new drawing. EnRoute supports the Multi Document Interface (MDI), which allows you to have more than one layout open at a time. |
|  | Open (Ctrl+O) - Use the Open command to open an existing EnRoute file. When you open a file it is placed in its own drawing window, not the current one. |
|  | Import a File - Use the Import command to load a file that has been saved in a compatible format other than the native EnRoute format. |
|  | Save (Ctrl+S) - Use the Save command to save the current design file. If you are saving a file for the first time, the Save File As dialog will be displayed. |
|  | Print – The Print command gives you the opportunity to print the current screen view or drawing. |
|  | Calculator - This will activate the windows calculator tool. |
|  | Save Snapshot of Active View – This tool allows you to take a snapshot of the active view and save it to a file. |
|  | Measure Lengths- This tool opens a toolbar with access to several measurement tools. |
|  | Cut (Ctrl+X) – The Cut command allows you to remove items from the design file and place them in the Windows Clipboard so they can be “pasted” either back in the original file, or in a another design file. |
-



Copy (Ctrl+C) – The Copy command is similar to the cut command, except it places a copy of the selection into the clipboard.



Paste (Ctrl+V) – The Paste command takes information that is stored in the Windows Clipboard and places it in the currently active design file.



Delete – Use this tool to delete the selected objects.

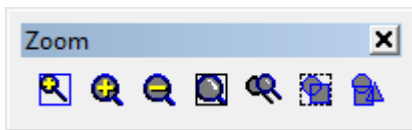


Undo (Ctrl+Z) - Use the Undo command to reverse your most recent actions. The Undo command may be chosen as many times as needed to undo previous editing operations up to the first operation in the editing session. If you load a file and make a number of editing changes to it, EnRoute will allow you to Undo all of the changes.



Redo (Shift+Ctrl+Z) Redo command to reverse any operation changed by the Undo command. You can continue to select the Redo command until all of the Undo operations have been reversed.

Zoom Toolbar



Zoom Window – Use the Zoom Window command to magnify a selected area. The Zoom Window command is different than the Zoom Out and Zoom In commands in that you can specify a particular area for the zoom to cover.



Zoom In (Ctrl+I) - Use the Zoom In command to zoom up or magnify the current window.



Zoom Out (Ctrl+K) - Use the Zoom Out command to zoom straight back.



Zoom to Plate (Ctrl+P) – Use the Zoom to Plate command to magnify to the extent of the plate size.



Zoom Previous - This command will return you to the previous zoom display.

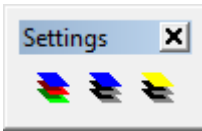


Zoom to Extents of Selection - Use the Zoom Selection command to display the selected group in the current layout.



Zoom to Extents of all Objects– Use the Zoom to the Extents command to display all design objects in the current file. This will include the plate if one is defined.

Settings Toolbar



Define Layers (F7) – This icon activates a dialog that allows the user to manage the layers that are used to store drawing elements. Within this dialog you can specify the active layer.

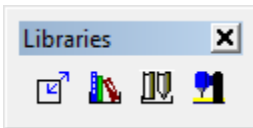


Change Layers – Activates a dialog that allows you to change the layer to store the current selection.



Toggle the layers toolbar – Opens or closes the layers toolbar.

Libraries Toolbar



Redefine the Current Plate – This activates the dialog to define the plate in the current drawing. The Plate provides a good reference during the design process and it also provides a means of making sure that toolpaths stay within the limits of the material you are using.



Material Library (F4) – The Material Library provides a method for customizing settings for your machine and specific activities. You may define speeds for different types of materials (wood, metal, etc.) that are to be routed or engraved. When the Define Plate command is used, there is the option to select a material type to be associated to the plate. Speed settings can be set for the materials you use most often so that EnRoute will apply these parameters to new toolpath strategies jobs without having to re-specify them each time.

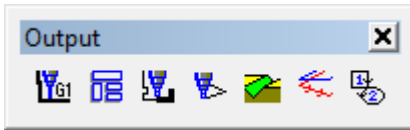


Tool Library (F5) - The Tool Library command is used to define routing and engraving tools used with the toolpath commands. The tool geometry entered for each tool is used to generate toolpaths.



Driver Setup (F6) - This icon will bring up the parameters for the currently active driver. Each driver parameter may be edited and then saved back to the driver configuration file. All of the drivers that have been installed on the computer are available to be selected as the active driver.

Output Toolbar



Generate Output - This is the main icon used to send toolpath output either to a file or directly to the machine. In this dialog the user defines how the information is to be sent, how the toolpaths are ordered as they are output, and which toolpaths are included.



Automatic Toolpath - The EnRoute Wood version of EnRoute provides Automatic Toolpath (ATP) capabilities that allow you to process parts contained in external DXF geometry files.



Simulate 2D (F8) – This icon activates a mode in which the user can visualize how the toolpaths will be sent to the machine. The toolpaths are drawn to the screen in the same sequence that they will be sent to the machine as output. Statistics regarding the number of tool changes and the number of tool lifts are also presented in this mode. In addition, the user has the opportunity to modify the order of the toolpaths and then immediately see the effect on the output using the 2D simulation.



Simulate Ortho - This icon activates a mode similar to the 2D Simulation except it is in the perspective view. The toolpaths are drawn to the screen in the same sequence that they will be sent to the machine as output. Statistics regarding the number of tool changes and the number of tool lifts are also presented in this mode. In addition, the user has the opportunity to modify the order of the toolpaths and then immediately see the effect on the output using the Simulate Ortho mode.



3D Rendered View - This icon activates a 3D mode that allows you to view a solid model of your material, and the results of the toolpaths cutting the material. This mode is useful for seeing the effects of both 2D and 3D toolpaths.



Project Toolpaths - This tool provides a method for tipping toolpaths vertically. It is useful for cutting non-horizontal surfaces, and is likely most useful for routing offset and fills.



Set Toolpath Order – This tool is used to manually set the order of the objects as they will be used to generate output. The user can define the order using different methods for sorting the objects or define the specific order by selecting the sequence with this tool.

Object Edit Toolbar



Selection Tool – This is the main selection tool in EnRoute for selecting, rotating and scaling objects.



Move Tool – This tool allows the user to precisely move an object.



Scale – This tool is used to precisely scale objects. By selecting a group of objects and then clicking on the scale icon the user can enter precise size or scale information to change the size of the selection.



Rotate – This tool is used to either interactively or precisely rotate selected objects.



Mirror Object - This tool is used to interactively mirror an object.



Group (Ctrl+G) – This command is used to group two or more objects together so they may be selected and manipulated as a unit. After the objects have been placed in a group, they can be selected as a group by clicking on any member of the group. Also, they may be released to return them to their original state as separate objects.



Ungroup (Ctrl+U) – This command is used to release objects from a group to their original state as separate objects.



Alignment – This icon can be used to activate a toolbar that presents a number of different options for aligning the objects in a selection. The first icon is used to align objects to center both horizontally and vertically. The additional icons in this toolbar provide other alignment options.



Weld –This icon is used to activate a toolbar that presents a number of options for welding selected objects. The first icon is used to weld objects that are overlapping each other into one single object. Additional options include: Weld Common and Weld Subtract, as well as two interactive welding tools.



Nest - This tool allows you to arrange selected objects into a configuration that will allow them to be cut out so they will efficiently utilize the material. Select the objects you want to nest, and then click on the Nest icon. This will activate a dialog for specifying gap and allowable rotation angles. Click OK, and the objects will be nested on the plate. This icon is also used to activate a toolbar that contains two additional methods of nesting, Dynamic Nesting and Multiple Nesting.

Toolpaths Toolbar



Edit Toolpath Plan – This icon activates the Plan dialog that allows you to edit the Plan associated with the selected toolpath groups. The Plan represents the strategies applied to a toolpath group.



Routing Offset – The Routing Offset command is used to offset, or compensate, for the engraving or routing tool. This will make sure no matter what size tool you use the parts will be the correct size. You can also use this command to create inlaid objects. Click and hold to open the flyout menu.



Hatch Fill – The Hatch Fill command is used to generate a series of toolpaths to remove an area or section of material.



Island Fill – The Island Fill option is a series of inlines (lines and arcs) that are generated until the selected contours are filled.



Drill - The drill functions provide tools for adding drill objects to the design. Drill objects are a special type of toolpath group that define drill points that can either be created on their own, or be created based on selected contours. Drill points can be created in patterns such as arrays, around a circle, and along a path. Multiple tools can be specified for any drill point.



Slot – The Slot command is used to create a single line that the router will cut.



Engrave – The Engrave command is used to generate a machine path on the original contour to be engraved or to generate a 3D path without a fill.

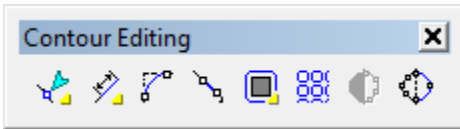


Pyramid – This type of toolpath strategy is used to create a positive bevel on the selected set of contours using a conic-shaped tool. It can also include a roughing pass to ensure that the pyramid tool will be able to complete its cuts, and an offset toolpath to cut the pieces out after the beveled tool has completed its work. This method is popular for making lettering and artwork that is commonly referred to as beveled, chiseled, sculpted, or prismatic.



Delete Toolpath – This command allows the user to delete toolpaths that have been applied to selected objects.

Contour Editing Toolbar



Edit Points - This tool provides access to a number of editing functions. You can move, insert, and delete points, open and close contours, change segments to arcs, curves and lines. After selecting the tool and clicking on a contour, right click to make these functions available.



Extend/Trim - The Extend tool is used to extend an open contour either interactively, by a specific distance, or to intersection with another contour segment. If you click and hold this button, a flyout menu will open. From there you can access the Trim tool.



Fillet Corners – This tool provides a method to insert rounded corners, or fillets, at selected corners in a contour.



Join Contours - This tool allows you to connect open contours at their endpoints either into an open or closed contour.



Offset Contours - This tool opens a dialog that allows you to enter parameters to offset a closed or an open contour.



Multicopy – This tool opens a dialog which allows you to produce multicopies of an object in three different ways.

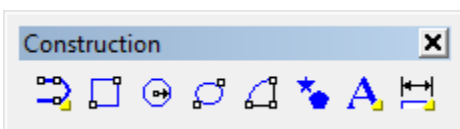


Vectorize Bitmap - This powerful tool is used to create a series of closed vector contours from a bitmap image. In order to create toolpaths in EnRoute, it is necessary for the artwork to be represented by vectors. This tool provides the means of converting scanned images or other bitmap images into vectors so that they may be used to create toolpaths.



Clean up Contours - When bitmap images are vectorized, the vectors often require some degree of 'cleaning up' in order for them to be appropriate to toolpath. This tool provides an automated means of cleaning up contours, by reducing the number of curve segments to create smooth contours that will then create better toolpaths.

Construction Toolbar



Draw Polyarc – This tool is used to create a series of connected line or arc segments that can either be open or closed. Click and hold to activate a flyout toolbar that contains a number of other line drawing tools.



Draw Rectangle – This tool will open the Precision Toolbar for the rectangle. Rectangles can be precisely or interactively created.



Draw Circle – This icon activates the Precision Toolbar for the circle. Several tools are available to interactively or precisely create a circle.



Draw Ellipse – This tool will open the Precision Toolbar for the ellipse. Create an ellipse interactively or precisely.



Draw Arc – This tool activates the Precision Toolbar that provides other arc construction options.



Draw Polygon – This button activates a dialog for defining the parameters for placing polygon shapes.

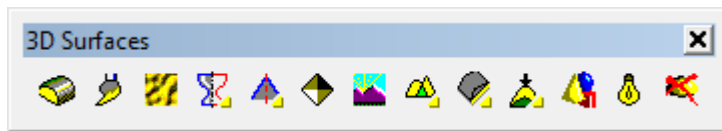


Text – This tool opens a Precision Toolbar. Any True Type® font that is installed on your computer is available to be used. The toolbar allows for many options in placing and editing the text to create the desired effects.



Create Dimensions – Click and hold to open a set of dimension tools to be used to measure and record in the drawing.

3D Surfaces Toolbar



Create Relief - This icon opens a dialog that enables several options to create a relief for the selected contours.



Edit Selected Relief – This icon opens a dialog that provides several relief editing tools that allow the user to interactively edit a selected relief.



Textures – Click and hold this icon to open a flyout menu containing several different textures that can be edited and applied to a relief.



Extrusions - This icon activates a toolbar that provides additional options in creating reliefs. These tools use revolutions and extrusions to create meshed objects and modify selected relief surfaces.



Chamfer - This icon activates a toolbar allowing the selection of different chamfering tools. The Chamfer tools allow you to modify reliefs by applying profiles that can range anywhere from a simple rounded edge all the way to an intricate profile that changes the whole relief.



Apply Mesh - This icon activates a dialog designed for the application of 3D meshes to reliefs. Meshes are made up of triangles of varying sizes that define their surface.



Apply Bitmap - This icon enables a dialog used to apply bitmap images as a method for modifying relief surfaces. The application method defines how the bitmap will be applied to the relief surface.



Merge Reliefs - This tool allows you to merge one or more reliefs with a base relief. This allows you to create portions of a relief separately and then merge them together to create the final surface.



Modify Relief – Click and hold this icon to activate a toolbar that provides tool options that allow you to modify reliefs to fit the needs of your design.



Align Reliefs - This icon can be used to activate a toolbar that presents a number of different options for aligning the reliefs in a selection. The first icon is used to align all reliefs to the bottom of the plate. The additional icons in this toolbar provide other alignment options.



Create Primitive Objects - The Primitives tool is used to create a range of basic 3D mesh shapes. These shapes can be used to build more complex shapes, or they can be used as design elements in your relief.

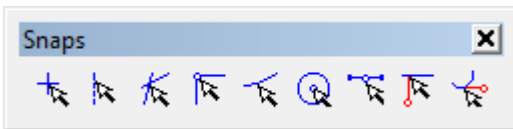


Rendered View - This icon activates a rendered view of the selected relief.



Delete Relief - Deletes the selected relief.

Snaps Toolbar



Snap to grid – Snaps to the nearest grid point. The grid spacing is configured on the Grid page in Preferences.



Snap to guideline – Snaps to a nearby guideline when the cursor gets close. Also when dragging an object, the perimeter of the object snaps to nearby guidelines.



Snap to intersection – Snaps to the intersection of two line segments.



Snap to endpoint – Snaps to the endpoints of arc, curve and sine segments.



Snap to nearby contour – Snaps to the nearest point on a contour when the cursor gets close to it.



Snap to center of arc – Snaps to the center of an arc segment when the cursor is placed near the arc.



Snap to midpoint – Snaps to the midpoint of a segment when the cursor gets close to it.

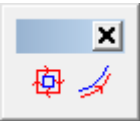



Snap to perpendicular point – When constructing line segments, after defining the first point in the segment, the second endpoint will snap to a point on a contour so that the line segment will be perpendicular to the contour.




Snap to tangent point – When constructing line segments, the second endpoint will snap to a point on an arc so that the new line segment will be tangent to the arc.

Routing Offset Toolbar





 **Routing Offset** – The Routing Offset command is used to offset, or compensate for, the engraving or routing tool. This will make sure no matter what size tool you use; the parts will be the correct size.


 **Routing Offset Open Contour** - This command is similar to the standard routing offset tool except that it is used for open contours instead of closed contours. This imposes some limitations on the types of offsets that make sense with an open contour.


Drill Toolbar





 **Drill Point** - The Drill Point command is used to create a single hole or pierce point.

 **Drill Circle** - The Drill Circle command is used to create a pattern of drill points around the radius of a circle.

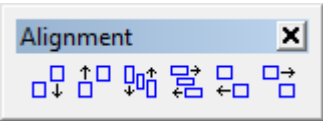
 **Drill Array** - The Drill Matrix command is used to create a series of drill points by rows and columns to form a grid.


 **Drill Center of Objects** -The Drill Centers command is used to place a drill point in the center of the selected contours.


 **Drill Along Contour** - The Drill Contours command is used to create a series of drill points along a contour.


 **Drill Corners of Plate** - The Drill Corners command is used to create a pattern of drill points around a Plate. You must have defined a plate before you can use this command.

Alignment Toolbar



 **Align Bottom** – Use this command to align the bottom of objects that have been selected.

 **Align Top** – Use this command to align the top of objects that have been selected.

 **Align Centers Vertical** - Use this command to align the vertical centers of selected objects.



Align Centers Horizontal – Use this command to align the horizontal centers of selected objects.

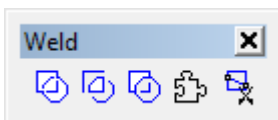


Align Left – Use this command to align selected objects to the left side.



Align Right – Use this command to align selected objects to the right side.

Weld Toolbar



Weld Joined – This is the most commonly used method for welding objects. It works by combining overlapping objects into one object by creating a new object that is defined by the outer perimeter of the overlapping objects.



Weld Common – This tool is used to define a new object based on overlapping contours. The new contour is defined by the perimeter of the overlapping portion of the contours.



Weld Subtract – Weld Subtract subtracts from the object that was selected first, using all subsequently selected objects.

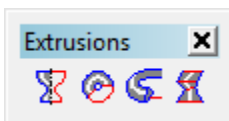


Jigsaw Weld - This is a design tool used to create a shape in which both open and closed contours may be used in the creation process. It is an interactive tool. By moving the cursor into different areas on the screen, EnRoute looks for closed shapes that can be formed by overlapping contours.



Cut by Line – The Cut by Line tool allows the user to “cut” an object with a line drawn across it to make two separate objects.

Extrusions Toolbar



Revolve to create surface - The Revolve tool allows you to use one or more contours to revolve around an axis to either modify a relief or to create a new mesh object.



Spin Contours - With the Spin tool the selected contours are spun around the Z-axis, located at a center of rotation that is defined by the user.

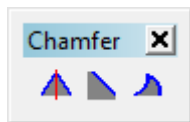


Extrude Contours - The Extrude tool provides a means to extrude a profile along a path to either modify a relief or create a mesh object.



Sweep Two Rails - The Sweep Two Rails tool provides another method of using profile contours to either modify an existing relief or create a new mesh surface.

Chamfer Toolbar



Chamfer Relief - The Standard Chamfer works by first adding a height to a relief that is equal to the height of the profile, and then removing material around the perimeter of the relief in the shape of the profile.

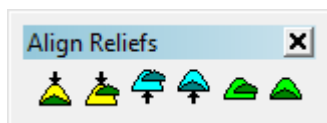


Chamfer Centerline - The Chamfer Centerline tool creates a chamfered surface using a profile where the profile size is modified so that it extends to the centerline of the shape.



Baroque Chamfer - This tool allows the user to simulate the effects of a traditional hand-carved surface. It automatically determines convex and concave sections of the artwork and then applies different profiles to these sections with smooth transitions between them.

Align Reliefs Toolbar



Align all reliefs to bottom of plate - Aligns each of the selected reliefs to the bottom of the plate.



Align selected reliefs to bottom of plate - Aligns the selection to the bottom of the plate. The relative position of the reliefs in the selection doesn't change.



Align all reliefs to top of plate - Aligns each of the selected reliefs to the top of the plate.



Align selected relief to top of plate - Aligns the selection to the top of the plate. The relative position of the reliefs in the selection doesn't change.

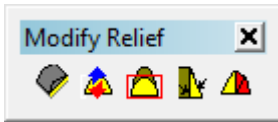


Align all reliefs to center of plate - Aligns each of the selected reliefs to the middle of the plate.



Align selected reliefs to center of plate - Aligns the selection to the middle of the plate. The relative position of the reliefs in the selection doesn't change.

Modify Relief Toolbar



Smooth Relief - This tool allows you to be able to soften, or smooth, the relief either to make it more attractive or to make it so that it will machine better.



Invert Relief - This is a simple way to convert a 'positive' relief to a 'negative' relief.



Fit Relief to Plate - This tool will automatically position and scale a relief so that it fits vertically within the defined plate in your active drawing.

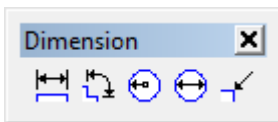


Apply Draft angle to Relief - This function allows you to add a draft angle to your relief.



Create a Mesh surface from Relief - This tool allows for the creation of a mesh surface from a relief.

Dimension Toolbar



Single Dimension – This tool allows you to determine the dimension between two selected points.



Angle Dimension – This tool allows you to determine the angle of two selected points.



Radius Dimension – This tool allows you to determine the radius of a selected circle.

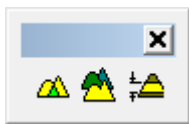


Diameter Dimension – This tool will determine the diameter of a selected contour.



Leader Dimension – This tool allows you to place a leader arrow in the selected location of the drawing.

Relief Toolbar



Merge Reliefs – This tool is used to merge one relief to another.

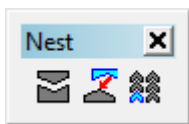


Combine Selected Reliefs – This tool will combine two or more selected reliefs.



Extract a slice from a relief – This tool allows for a section of the relief to be extracted from the relief.

Nest Toolbar



Nest– This tool is used to automatically nest the objects on the plate according to selected parameters.

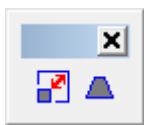


Dynamic Nest – This tool allows the user to specifically place the objects on the plate.



MultiNest – This tool will nest a specified number of objects according to the parameters entered in the precision toolbar.

Scale Toolbar



Scale – This tool is use to interactively scale an object.



Distort – This tool lets you stretch an object disproportionately.

Fill Toolbar

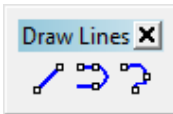


Island Fill – This tool is used to create Island Fill toolpaths.



Spiral Fill – This tool is used to create toolpaths in a spiral pattern.

Draw Lines Toolbar



Draw Line – This tool is used to draw a line between two specific points.



Draw Polyarc - The new Polyarc tool allows you to construct a series of lines and arcs to form a new contour.



Draw Bezier Curve – This tool allows you to draw precisely curved lines.

Extend/Trim Toolbar

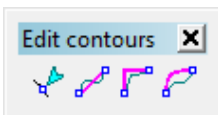


Extend Contour –This tool is used to extend an open contour either interactively, by a specific distance, or to and intersection with another contour segment.



Trim Contour – This tool allows you to shorten a contour either interactively, by a specific distance, or based on intersections with boundary contours.

Edit Contours Toolbar



Edit Points – This tool allows you to manipulate line, curves and points in contours.



Linearize Curve Section - This tool allows you to edit a contour to define line segments between two points.

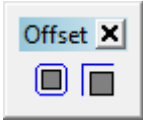


Insert Corner – This tool allows you to interactively place a corner in an existing contour.



Smooth Approximation Tool – This tool allows you to interactively edit a contour to smooth the segment between two selected points.

Offset Toolbar

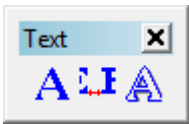


Offset Closed Contour – This tool allows you to define one or more offsets to a selected closed contour.



Offset Open Contour – This tool allows you to create offsets for selected open contours.

Text Toolbar



Create New Text Objects – This tool is used to interactively create text objects.

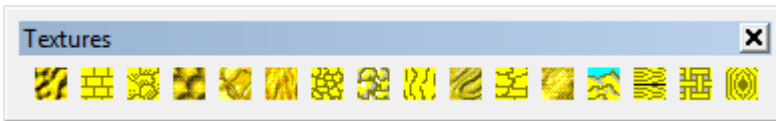


Edit Existing Text Objects – Use this tool to edit existing text objects.



Convert Text Objects to Curves – This tool converts the text objects to curves so that you can then apply toolpaths.

Textures Toolbar



Basic Noise Texture – This texture creates a bumpy surface using just a power parameter and the scale of the texture.



Brick Texture – This is a simple Bricks and Mortar texture that can be applied to reliefs.



Cellular Texture – Cellular Texture breaks up space into cells or regions.



Flagstone Texture – Flagstone texture simulates a mosaic of stone tiles. The stones appear to be selected to fit together but have a random grain direction.



Flame Texture – Flames texture simulates the look of flames.



Hammered Texture – This texture can create many different looks of textures.



Hexes Texture – Hexes texture produces a hexagonal or honeycomb pattern.



Marble Texture – Marble texture produces a pattern of veins in the relief.



Mudpot Texture – The Mudpot texture basically looks like the surface of bubbling mud.



MultiCell Texture – The MultiCell texture is related to the cellular texture but with more control parameters. Cellular texture breaks up space into cells or regions. The point being evaluated falls into one cell. The texture calculates the distance to the center of that cell and centers of adjoining cells. Combining these distances gives different effects.



MultiFract Texture – MultiFract texture can be generally described as a roughness texture. Adjusting the parameters will change the appearance of the texture in a wide variety of ways.



Terrain Texture – Terrain texture gives options to create the look of the landscape.



Veneer Texture – Veneer texture like the Wood texture simulates the look of wood rings and grain in the relief. The veneer produces a series of slices through the wood and tiles them next to each other.



Weave Texture – Weave texture produces a simple basket weave or textile pattern in the relief.



Wood Texture – Wood texture simulates the look of wood rings and grain in the relief. The appearance of different species of wood can be simulated by adjusting the various parameters of the texture.

Other Command Icons

This section identifies icons from many of the tools in EnRoute. The main intent here is to provide a listing of EnRoute icons in one place for your review. All of these functions are explained in more detail, within the context of the EnRoute functions to which they belong.



Add relief – This option is used to create a raised relief.



Apply Mesh to Relief – This tool allows you to apply a mesh object to a relief.



Arc by center and two points – Use this command to define an arc by defining its center and two endpoints. It is defined interactively; you can either leave it as an open contour with just the arc, or define it as a closed contour by placing the cursor either outside or inside the arc as it is constructed.



Arc by point and angle – Define an arc by entering coordinates for the center and start, and then defining the angle turned by the arc.



Arc by three points – This is a handy interactive tool to define an arc by clicking on the start point, then clicking on the end point, and then dragging to define an intermediate point on the curve.



Arc copy – In the Multicopy function, select this parameter to copy around an arc.



Array copy – Creates multiple copies of an object according to the selected parameters.



Bevel Relief – This tool is chosen to create a beveled relief.



Bevel Relief Edit Tool – This tool allows editing of a relief using a beveled shaped tool.



Blending Relief Edit Tool – This tool will blend the surface of the relief within the parameters specified in the toolbar dialog.



Box – Box shaped mesh found in create primitive objects.



Circle by Center and Point – This tool allows you to enter the coordinates for both the center and a point on the circle to define the circle.



Circle by Center and Radius – Use this tool to click on a circle center and then drag to define its size. You can also define the circle by coordinates and dimension.






















Circle by corners – Click and drag to define the corners of a boundary box around a circle, or define these corners by precision input.



Circle by three points – This is an interactive tool used to define a circle. Click on the first point then click, hold and drag to define the circle.



Concave polygon – Choose this tool to draw a concave polygon, or star shape.

	Cone – Cone shaped mesh found in create primitive objects.
	Convex Polygon – Choose this tool to draw a convex polygon.
	Corner chamfer – This tool will chamfer the corners of a rectangle. Specific parameters can be entered to determine the radius and placement of the rectangle.
	Corner fillet – This tool allows you to place a fillet on the corner of a rectangle to your desired parameters.
	Corner reverse fillet – This tool puts a reverse fillet on the corners of a rectangle.
	Corner square – When this option is chosen, the corners of the rectangle is square.
	Cylinder – Cylinder shaped mesh found in create primitive objects.
	Dimension Arrow Height - This parameter determines the height of the arrow.
	Dimension Arrow Width – This parameter determines the width of the arrow.
	Dimension Decimal position for arc – This parameter sets the number of decimal positions to be displayed for the arc dimensions.
	Dimension Decimal position for Line – This parameter sets the number of decimal positions to be displayed for the line dimensions.
	Dimension Extension line parameter - This parameter sets the distance that the extension lines start to draw from the start and end points of the dimension.
	Dimension parameter Height -This parameter sets the height of the text in the dimensions tool.
	Dimension Text Position above - This sets the position of the text above the arrow line.
	Dimension Text Position Center - This sets the position of the text in the center of the arrow line.
	Dimension Text Position Below - This sets the position of the text under the arrow line.
	Disc – Disc shaped mesh found in create primitive objects.
	Dome Relief Edit Tool – This tool is used to edit a relief in a dome shape.
	Edit Bridges – The edit bridges tool allows you to edit individual bridges, moving them and changing their parameters.



Edit entry exit – Each cut in a toolpath has its own entry and exit points. Use this tool to edit the points or type of entry and exit.



Ellipse by dimensions – Enter the coordinates for the center of an ellipse and then define its height and width along its two axes.



Ellipse from corner to corner – This tool is similar to the circle command of this same type. Just interactively click and drag on the corners of a boundary rectangle to define an ellipse.



Explode selected contours – This tool will separate a selected contour into individual segments.



Faceted mesh to relief – After a mesh is created, it is then added to a relief. If the faceted option is selected, then the object is applied to the relief surface using its facets directly.



Flat relief – This tool is chosen to create a flat relief.



Flat Relief Edit Tool – This tool is used to edit a relief using a flat shape.



Horizontal Distance Multicopy – This tool defines the distance horizontally between the objects when using the Multicopy function.



Horizontal Distance Multicopy - This tool defines the distance horizontally between objects in Multicopy from the front end of an object to the front end of the next object.



Measure Tool Angle – This tool will calculate the degree of the angle between two selected points on a contour, or three selected points not on a contour.



Measure Tool Area – This tool will calculate the area of a closed contour. If you select more than one contour, it will also add the total area of the selected objects.



Measure Tool Segment - This tool will measure the distance between two points. It will continue to add each selected segment to the total.



Merge highest relief – This option is used to modify a relief. The new portion of the relief only modifies the existing relief where it is higher than the existing relief.



Merge lowest relief – This option is similar to the Merge Highest option, except that a relief is only modified where the new relief is lower than the existing relief.



Merge selection – The Merge Selection tool is used to join open contours together based on a definable tolerance.





















Mesh – Mesh objects are made up of triangles of varying sizes that define their surface.





















Mirror Horizontal – This tool reverses the selected contour so that it becomes its mirror image horizontally.



Mirror Vertical – This tool reverses the selected contour so that it becomes its mirror image vertically.

	Move Absolute - Absolute move will move the selected object from one point to another precisely or interactively.
	Move Corner - The Corner move tool allows you to move the selection based on the points of the bounding box for the selection.
	Move Relative - The Relative move allows the user to move the selection by a specific amount in each direction.
	Move Text - Select this tool to move the location of the selected text up or down. A positive value will move up; a negative value will move down.
	Multiple Fillet – This tool is used to fillet multiple corners of a selected object.
	Path copy – In the Multicopy function this will allow you to create multiple copies along a path specified.
	Plane – Plane shaped mesh found in Create Primitive Objects.
	Project toolpaths to plane – Use this tool to change the height or angle of existing toolpaths.
	Rectangle by dimensions – Enter the parameters to define a corner of a rectangle and then specify its size by height and width.
	Rectangle corner to corner - Use this tool to define a rectangle by clicking on one corner and dragging to define to the opposite corner. Hold the shift key down to keep the rectangle a square.
	Replace relief – This option is also used primarily to modify an existing relief. The applied profile takes the place of the existing relief wherever the new profile overlaps the existing profile.
	Reset Parameters – This tool is used to reset the ordering parameters of the toolpaths using the Preferences.
	Reverse Open Contours – This tool will reverse the direction of the selected open contour.
	Rounded relief – This tool is chosen to create a rounded relief.
	Save Parameters – This tool will save the current parameters to the preferences.
	Scale 3 Dimensions – In the scale tool, this option will scale the selection in all 3 axes, based on the scale factors that are defined either precisely or interactively.
	Scale 2 Dimensions – In the scale tool, this option allows three-dimensional objects such as reliefs and meshes to be scaled in just two dimensions. For example, you can scale a relief in XY dimensions without changing its thickness.
	Scale 1 Dimension – In the scale tool, this option allows you to scale a selection in just 1 direction. Stretch the x-dimension without changing the y or z size of a selection.

	Set start point – Use this option in the edit points tool to set the start point of a contour.
	Simulate done – Close the simulation.
	Simulate Draw Next Segment – The start of the simulation.
	Simulate next tool lift – This will play the simulation until the tool is lifted.
	Simulate pause – This will stop the simulation.
	Simulate to end of job – This will play the simulation to the end of the job.
	Simulate to tool change – This will play the simulation until the tool is finished.
	Simulate rewind – In the simulation tool, this is the function to rewind the simulation to play it again.
	Single Fillet – This tool is used to interactively place a fillet into a select area of a contour.
	Slant Text - Select this option to change the slant of the characters. A positive value will cause the letter to lean towards the right, while a negative value will make the letter lean back or to the left.
	Smooth mesh to relief – This option implements a means of smoothing out the facets of the mesh to provide a smoother finished surface when applying the mesh to the relief.
	Smooth Relief Edit Tool – This tool has a smooth shape to use when editing the existing relief.
	Sphere – Sphere shaped mesh found in create primitive objects.
	Subtract relief – This option is used to create a recessed relief.
	Text Height - Select this tool to change the height of the text objects.
	Text position on contour - Select this tool to determine which side of a line or curve the selected text is to be placed.
	Text Spacing - Select this tool to change the spacing between the text objects. A positive value will increase the spacing; a negative value will decrease the spacing between characters.
	Text Width - Select this tool to change the width of the selected text objects.



Texture Relief Edit Tool – This tool allows the user to edit a relief with a texture tool.



Torus - Torus shaped mesh found in create primitive objects.



Vertical distance Multicopy – This tool defines the distance vertically between objects when using the Multicopy tool.




Vertical distance Multicopy – This tool defines the distance vertically between the objects in Multicopy from the front end of an object to the front end of the next object.

4. Working with Files


Creating New Files

Menu: File / New

Toolbar: File / New

Click **New**.  The keyboard shortcut is Ctrl+ N.

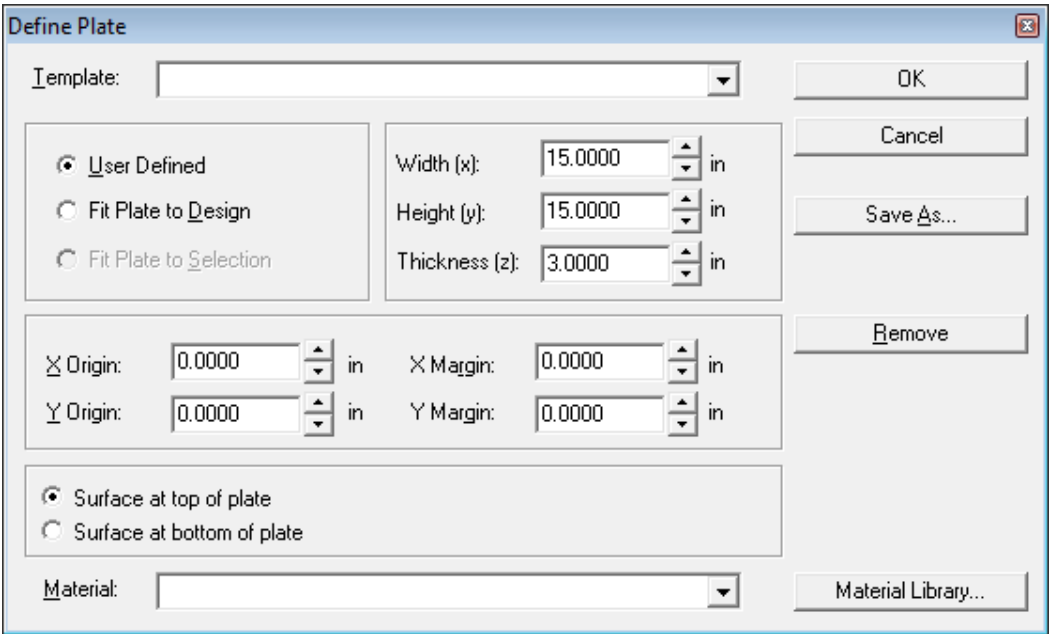
EnRoute supports the Multi Document Interface (MDI), which allows you to have more than one design open at a time.

 If you have the **Prompt for Plate** option turned **ON** (see “Setting Preferences”), you will be asked to define the Plate each time you start a new design.

Defining a Plate

Menu: Machining / Define Plate

Toolbar: Libraries / Redefine the current plate



The **Define Plate** dialog box is used to specify the dimensions and material of the plate. It includes the following fields and options:

- Template:** A dropdown menu.
- Options:** Three radio buttons: ☒ **User Defined**, ☐ **Fit Plate to Design**, and ☐ **Fit Plate to Selection**.
- Dimensions:** Four input fields with spinners: **Width (x):** 15.0000 in, **Height (y):** 15.0000 in, **Thickness (z):** 3.0000 in, **X Origin:** 0.0000 in, **Y Origin:** 0.0000 in, **X Margin:** 0.0000 in, and **Y Margin:** 0.0000 in.
- Surface:** Two radio buttons: ☒ **Surface at top of plate** and ☐ **Surface at bottom of plate**.
- Material:** A dropdown menu.
- Buttons:** **OK**, **Cancel**, **Save As...**, **Remove**, and **Material Library...**.

When a new design is created, the Define Plate dialog opens.

The Define Plate dialog allows you to specify the dimensions of the material on which you are working, as well as the type of material.

The following options are available:

User Defined	If this option is selected, you can enter the Origin, Width and Height of the material to be used.
Fit Plate to Design	If this option is selected, a plate will automatically be created with height and width values based on the current design.
Fit Plate to Selection	If this option is selected, a plate will be created based on the currently selected contours.
Width	The measurement of the plate along the X-axis.
Height	The measurement of the plate along the Y-axis.
Thickness	The measurement of the plate along the Z-axis.
X/Y Origin	The X/Y Origin is the position of the lower left corner of the plate.
X/Y Margin	If a margin is entered, a second, dashed rectangle is created on the inside of the plate's edges. The margin is used to help create the layout. It can be very helpful for masking out the locations of clamps.
Surface Option	Surface at top of plate tells the program that you have set the surface at the top of the material. If you choose surface at bottom of plate, this means that you will set the surface at the top of the table (or bottom of the material).
Material	Clicking on the Material list box will display the types of materials (wood, metal, etc.) that are defined. Selecting the Material Library button will allow you to create a new material type, and the routing speeds associated with that material.
OK	Click OK to accept.
Cancel	Click Cancel to close this dialog.
Save As	Click Save As to save the current parameters as a template. The parameters can then be reloaded from the library by selecting the template from the Template list.
Remove	Click to remove the plate that is currently defined. You will be returned to the layout window and the plate will be removed.
Material Library	The Material Library may be accessed from this dialog. This allows you to define new material, change information on previously defined material, and save new or edited information set for specific materials. See "Editing the Material Library" for details.

Creating a Basic Plate



1. Click the **Define Plate** icon.
2. Enter the **Width** (distance along the X axis).
3. Enter the **Height** (distance along the Y axis).
4. Enter the **Thickness** of the material.
5. Enter values for **Margin** and **Origin** if desired.
6. Choose a **Material Type**. (This is optional).

7. Check to **surface at the top** or **bottom** of plate.
8. Click **OK** to create the plate.

Using Templates

Templates are used to store frequently used settings. Selecting the template reloads the information saved in the template.

Saving a Template


1. Enter the desired information in the Define Plate dialog box.
2. Click **Save As**.
3. Enter the name of this template and click **OK**.
4. This template will now be added to your plate template library.

Using a Template

1. Click on the **Template** list and the saved plate templates will be displayed.
2. Select the desired template.


Opening a Design


Menu: File / Open

1. Click **Open**  or **Ctrl +O**.
2. Select the file format, folder and the file that will be opened.
3. Press **Enter** or click **Open**.

Saving a Design

Menu: File / Save

1. From the **File** menu, select **Save** or **Save as**. 
2. If you have selected the **Save as** command or you are saving the document for the first time, a dialog box is displayed. Enter the name and the location that will be used to save the document.
3. Click **Save**.

 The file will be saved in the currently active drive and folder. Be sure that you know where the file is being saved so that you will be able to locate it the next time you want to open it.


Closing a Design



Menu: File / Close

1. From the **File** menu, select **Close**.
2. If your file has changed since the last time it was saved, a dialog box is displayed asking if you want to save the current file.
 - Click **Yes** to save the file before closing it.
 - Click **No** to close the file without saving the changes.
 - Click **Cancel** to keep the file open.

Importing a Design

Menu: File / Import.


1. Click **Import**. 
2. Select the file format, folder and the file that will be imported.
3. Press **Enter** or click **Open**.


-  See the Preference command located in the Setup menu for options regarding the Import command.
-  Bitmap graphics must be converted into contours before they can be used to generate toolpaths. See "Vectorizing Bitmaps" on for details.

Exporting a Design

Menu: File / Export

The software can export a design to a number of different 2D drawing formats.


-  Only the 2D design will be exported. Toolpaths will not be exported, and the software cannot export a 3D model of a finished piece.

1. Click **Export**. 
2. Select the file format, folder and the file name.
3. Type the name of the file to be exported in the File Name box.
4. Select the directory and drive you want the file exported to.
5. Press **Enter** or click **Save**.

Printing

Menu: File / Print

Use the Print command to print the active design.


1. Click **Print**.  This will open the Print dialog.
2. Select the type of print option.
 - Design** Select this option to send all the contours that exist in the current layout to the printer.
 - Selection** Select this option to send only the selected contours to the printer.
 - Window** Select this option to send the layout window to the printer. Exactly what is displayed on screen is what is sent to the printer.
 - Plate** Select this option to print only the plate.
3. Click **OK**.

Setup

Click the setup button to open the print setup dialog.

Calculator


Menu: File / Calculator

If you find the need for a calculator, EnRoute provides quick access to the Windows calculator with a simple click on the calculator icon. 

Snapshot

Menu: File / Snapshot

EnRoute has the ability to take a snapshot of the active view.

1. Click on the **Snapshot** icon. 
2. Select the file type and location that you wish to save the file.
3. Click the **Save** button.

The file format types that are available are listed here:

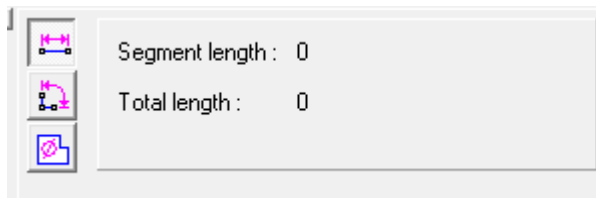
- AI
- DXF
- Flexi 5.8
- HPGL
- CasMate
- Stereolithograph
- JPEG Image

Measure

Menu: File / Measure



The Measure tool allows you to easily calculate angles, segments and area of an object.




Segment Length - This tool will measure the distance between two points. It will continue to add each selected segment to the total.

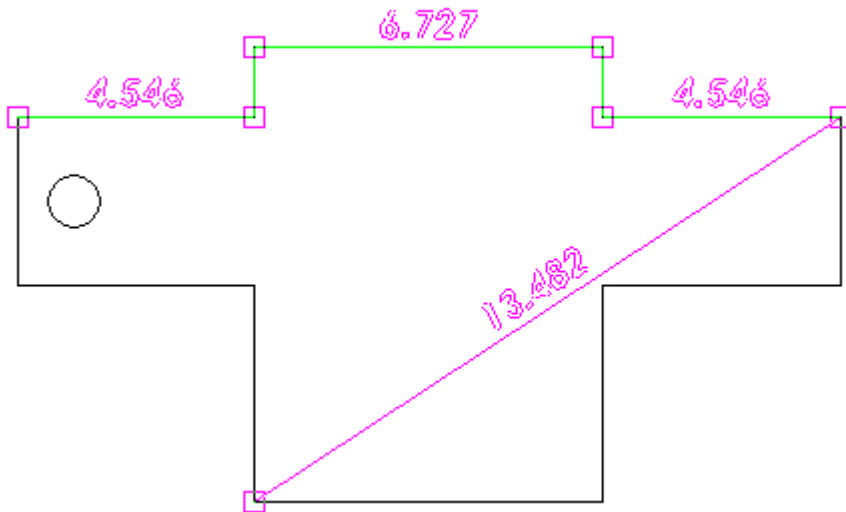


Angle Measure Tool - This tool will calculate the degree of the angle between two selected points on a contour, or three selected points not on a contour.



Area Measure Tool - This tool will calculate the area of a closed contour. If you select more than one contour, it will also add the total area of the selected objects.

1. Click on the **Measure tool** to activate the Precision Toolbar. 
2. Select the icon for measurement type: Segment, Angle or Area.
3. Using the snap tools will help you to easily select specific points in order to get accurate measurements.
4. The Precision Toolbar displays the information for the function that you are performing. In addition, temporary dimensions are displayed showing the measurement.
5. Right click in order to reset the measurement value. If you click for measurements without resetting, the new measurements will continue to be displayed. Right click any time to reset and clear the displayed measurements.
6. Changing from a length measurement to an angle measurement also resets the display.



5. Creating Contours

A selection of drawing tools is provided to allow you to create your design elements. Design elements may be created interactively by clicking and dragging, or they may be created precisely by entering desired values in the Precision Toolbars.



Line, Polyarc or Bezier Curve


Drawing a Line

Menu: Draw / Draw Line

Toolbar: Construction / Draw Polyarc / Draw Line

The Line tool draws a straight line segment connecting two points.

1. Click and hold **Polyarc**  then select **Draw** line. 
2. Click at the desired starting point then move the cursor to where you want the line to end and then click again.

 Hold the **SHIFT** key down while drawing to force the tool to create only horizontal and vertical lines.

You can also press the **F2** key to enter each starting and ending point's coordinates into the Precision Input Center. Enter each point's coordinates into the **X**, **Y** and **Z** fields and click **OK**.

Polyarc

Menu: Draw / Draw Line

Toolbar: Construction / Draw Polyarc / Draw Polyarc

The new Polyarc tool allows you to construct a series of lines and arcs to form a new contour. Although it is always possible to construct lines and arcs as separate entities and then join them together to form a contour, this tool allows you to accomplish both tasks together.

After activating the Polyarc command, the Precision Toolbar is activated, allowing you to choose between line and arc mode. The following table defines the buttons in the Precision Toolbar that present the available options for constructing lines and arcs.



Line - This toggles the segment creation mode to the line mode.



Arc - This toggles the segment creation mode to the arc mode.



Position - This allows you to define a position for either the start of a segment or the end of the active segment.



Direction – This allows you to define a direction for either the line or arc you are creating. If you are defining a line, the direction is the direction of the line. If you are defining an arc, the direction is the direction of the tangent to the arc relative to the previous segment. If it is the first segment, then the direction is relative to a line to the right.



Arc Center – This is only available in arc mode and is the center of the arc you are creating.



Radius – This is the length of the radius of the arc you are creating.



Sweep Angle – This is the sweep angle of the arc you are creating.

As you use the tool, you see that only the options that are possible based on where you are in the command are available at any given time. If you are in line mode then the arc parameters are disabled.

The idea with this tool is to construct segments, one at a time, and to use the method that fits the information you have available. Construction of the segments can be done either interactively or by entering values into the available edit boxes in the Precision Toolbar. This tool was designed to be used exclusively in the top view because you are creating arcs, and EnRoute expects arcs to be planar in the x-y plane.

Constructing a Line Segment – following are the simple steps required to create a line segment.



Activate the Line Mode.



If this segment is the first segment in you Polyarc contour, then either snap/click to the correct location in the drawing or define the correct x- and y-coordinates in the toolbar in order to start the segment.



Then define the next position for the endpoint of the line.



The other option for the endpoint of the line is to switch to the direction mode and then define an angle and distance in order to complete the line segment.



You are now ready for the next segment.

Constructing an Arc Segment – following are the steps required to create an arc segment. With an arc segment, EnRoute is trying to get enough information to define the arc. As soon as enough information has been defined then the arc is created. You can use many different combinations of information – start point, direction, radius, center point, sweep angle, and end point – to define the arc. The key is to remember that you are always trying to define enough parameters to allow the arc to be finalized.

The following examples demonstrate how arcs can be constructed in different ways.

Example 1



Activate the Arc Mode.



If this segment is the first segment in your Polyarc contour, then either snap/click to the correct location in the drawing or define the correct x- and y-coordinates in the toolbar in order to start the segment.



Then define the next position for the endpoint of the arc. The endpoint allows the arc to be constructed because the arc is assumed to be tangent to the previous segment. If it's the first segment then the tangent is assumed to be 0 degrees. With the endpoint and the tangent known, the rest of the arc parameters can be calculated and the arc constructed.



You are now ready for the next segment.

Example 2



Activate the Arc Mode if necessary.



If this segment is the first segment in your Polyarc contour, then either snap/click to the correct location in the drawing or define the correct x- and y-coordinates in the toolbar in order to start the segment.



Now switch to direction. You can now either define the direction by clicking on a point or by defining the direction in the edit box.



The position mode is activated, so just click on a point or define its coordinates in order to complete the arc.



You are now ready for the next segment.

Example 3



Activate the Arc Mode if necessary.



If this segment is the first segment in your Polyarc contour, then either snap/click to the correct location in the drawing or define the correct x- and y-coordinates in the toolbar in order to start the segment.



Now switch to center point. You can now either define the center of the arc either by clicking on a point or by defining the point in the edit boxes.



Now you just need to define either the endpoint of the arc or the sweep angle in order to complete this segment. Choose the preferred mode and then either enter the information in the toolbar or click on the appropriate point in the drawing. This will complete the segment.

Or





You are now ready for the next segment.

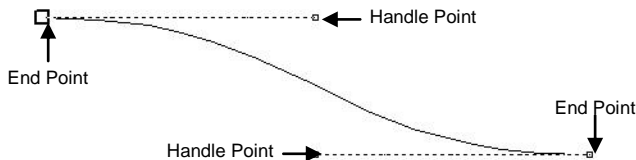
If you try these examples in your drawing, it demonstrates how arcs can be constructed very precisely by providing just enough information to construct each segment. When you are constructing precise parts that are based either on customer specifications, or on a drawing that you have been provided, this tool allows you to work your way around the part using the information available for each segment.

Bezier Curve

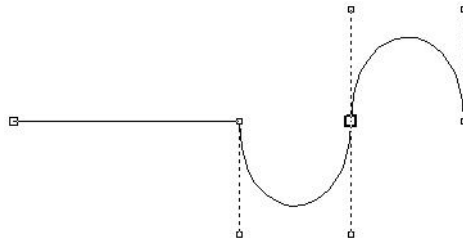
Menu: Draw / Draw Line



Toolbar: Construction / Draw Polyarc / Draw Bezier Curve


Bezier curves provide a way of drawing precisely curved lines on a computer. Each section of a Bezier curve is defined by four points: its beginning and end points, and two handle points which determine the shape of the curve.



Normally, a Bezier curve is composed of multiple segments. Each segment can be a curve or a straight line.





1. Click and hold  then click **Draw Bezier curve.** 
2. To place the first point in the curve, do one of the following:
3. Click once without dragging to place the point. This will make the first segment a straight line.
4. Click to place the starting point and drag to create its handle. This will make the first segment a curve.
5. For all subsequent points in the curve, click to place the point and drag to create its handle. If you do not drag to create a handle, the next line segment will be a straight line.
6. Right-click to release the tool and end the curve.

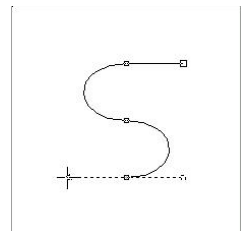
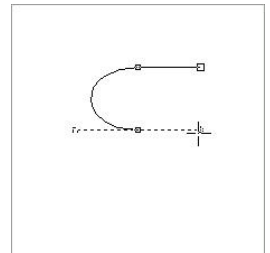
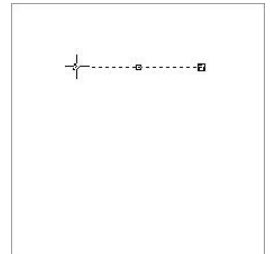
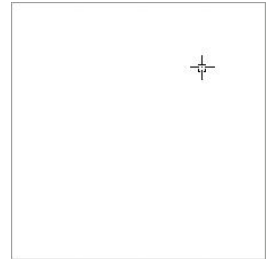
-  If one of your segments does not come out exactly right, keep going and finish the curve: you can easily come back and edit the segments later. See “Editing Points on a Curve or Contour” for details.

You cannot create a Bezier curve using the Precision Input Center. You must create the curves with the mouse.

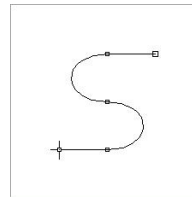
Example:

To draw a simple s-curve:

1. Click and hold  then click **Draw Bezier curve**. 
2. Click to create the upper right-hand starting point of the curve.
3. Click one grid unit to the left of the first point and drag one additional point to the left. This sets you up for the creation of the first curve.
4. Click one grid unit below the second point and drag one unit to the right.
5. Click one grid unit below the third point and drag one unit to the left.



- Click one grid unit to the left of the fourth point and right-click to finish drawing the curve.




Rectangle

The rectangle tool was designed to allow the user to construct rectangles using whatever method is most convenient. All of these construction methods are available within one tool, and you can switch between them at any time. Also, you can define rectangles either interactively or precisely, and you can combine both methods. Following are some specific instructions for creating rectangles.

Drawing a Rectangle by Corners


Menu: Draw / Rectangle

Toolbar: Construction / Rectangle / Corner to Corner

- Click the **Rectangle Icon**. This will open the Precision Toolbar. Select the **Draw from Corner to Corner**. 
- Click to place the first corner, and then drag the mouse to the location of the opposite corner and click the mouse again to place the rectangle.
- If desired, you can continue to create more rectangles. Repeat step 2 to place the rectangles.
- Right-click to end the rectangle function.

 Hold down the **SHIFT** key to force the software to draw only squares.

You can also enter each corner's coordinates into the **X** and **Y** fields and click **Apply**.


 The **Corner** icons in the Precision Toolbar allow you to create a rectangle with radiused corners. Select the style of corner you want and the size of the radius.

Rectangle by Dimensions


Menu: Draw / Rectangle

Toolbar: Construction / Rectangle / By Dimensions

To draw a rectangle by defining a starting point and the height and width of the rectangle:

- Click the **Rectangle Icon**. This will bring up the **Precision Toolbar**. Select **Draw by Dimensions**. 

2. In the Precision Toolbar, click the reference grid button that corresponds to the corner, side or center point that will be used to position the rectangle.
3. Enter the **Height** and **Width** values to define the size of the rectangle.
4. Left click to place the rectangle or enter the X, and Y coordinates for the positioning point.
5. To create a rectangle with radiused corners, select the style of corner you want and the size of the radius.

 The selected corner option will only be constructed if both the height and width of the rectangle are large enough to allow the size (radius) of the corner.

6. Click **Apply**. The rectangle will be placed in the workspace.
7. If desired, you can continue to draw more rectangles. Left click to place additional rectangles with the same values or enter the new values for the rectangles in the fields as before and click **Apply** to draw.
8. Right-click to end the rectangle function.


Circle


Like the rectangle tool, the circle tool provides several options for defining a circle. All of the options are available in one tool, and you can define coordinates either interactively or precisely.

Drawing a Circle by Center and Radius:

Menu: Draw / Circle

Toolbar: Construction / Circle / By Center and Radius


1. Click the **Circle Icon**. From the Precision Toolbar select **By Center and Radius**. 
2. Enter the value for the Radius of the circle. Left click the mouse to place the circle.
3. If desired, you can continue to create more circles of the same size by left clicking the mouse to place the additional circles. The circles can also be precisely placed by entering the x and y coordinates for the center of the circle and click **Apply**.
4. Right-click to end the circle function.


 After defining the radius for a circle, you can place multiple circles with that radius just by clicking to define center locations.

Drawing a Circle by Center and Point

Menu: Draw / Circle

Toolbar: Construction / Circle / By Center and Point


1. Click the **Circle Icon**. This will bring up the Precision Toolbar. Select **Circle by Center and point**. 
2. Click to place the center point of the circle, then drag to define the point on its radius. Click the mouse again to complete the circle.
3. If desired, you can continue to create more circles. Repeat step 2.
4. Right-click to end the circle function.
5. You can also enter the data for the center and the point into the corresponding fields. Click **Apply**.

 Don't forget that you can use snaps to precisely locate the points on your circle.

Drawing a Circle by Three Points

Menu: Draw / Circle

Toolbar: Construction / Circle / By Three Points


1. Click the **Circle Icon**. This will open the Precision Toolbar. Select **By Three Points**. 
2. Click to place the first point on the circle.
3. Click to place the center point on the circle, then drag to define the third point. Click the mouse again to complete the circle.
4. If desired, you can continue to create more circles. Repeat steps 2 and 3.
5. Right-click to end the circle function.

You can also enter the data for the points into the corresponding **X** and **Y** fields. Click **Apply**.

Drawing a Circle by Corners

Menu: Draw / Circle

Toolbar: Construction / Circle / Circle by Corners

1. Click the **Circle Icon**. This will open the Precision Toolbar. Select **Circle by Corners**. 
2. Click to place the first corner, then drag to define the opposite corner. Click the mouse again to complete the circle.
3. If desired, you can continue to create more circles. Repeat step 2.
4. Right-click to end the circle function.

You can also enter the **X** and **Y** coordinates for the corners and click **Apply**.


Arc

The arc tool provides three different methods for constructing an arc. All of the methods allow both interactive and precise definition of points and parameters. After the arc is constructed, it is a separate open contour. If your design is comprised of several open contours before certain types of toolpaths, such as fills, may be applied.

Drawing an Arc by Center, Start and Finish

Menu: Draw / Arc

Toolbar: Construction / Arc / By Center, Start and Finish


1. Click the Arc **Icon**. This will open the Precision Toolbar. Select Arc by Center, Start and Finish. 
2. Click to define the arc center.
3. Click to define the start point.
4. Click to define the arc endpoint.
5. If desired, you can continue to create more arcs using the same method.
6. Right-click to end the arc function.

You can also enter the data into the **X** and **Y** fields for each point and click **Apply**.

Drawing an Arc by Center, Radius and Angle

Menu: Draw / Arc

Toolbar: Construction / Arc / By Center, Radius and Angle


1. Click the Arc **Icon**. This will open the Precision Toolbar. Select **Arc by Center, Radius and Angle**. 
2. Enter the X and Y coordinates for the Center point.
3. Enter the Radius parameter.
4. Enter the Start Angle parameter
5. Enter the Sweep angle parameter. Left click to place the arc. Left click to place more arcs with the same parameters.
6. If desired, you can continue to create more arcs using the same method.
7. Right-click to end the arc function.

You can also enter the data into the **X** and **Y** fields for the points and enter the **degree** of the angle and click **Apply**.

Drawing an Arc by Three Points

Menu: Draw / Arc

Toolbar: Construction / Arc / By Three Points

1. Click the **Arc Icon**. The Precision Toolbar will open. Select **By Three Points**. 
2. Click to define the start point.
3. Click to define the end point.
4. Click or drag to define a third point along the arc.
5. If desired, you can continue to create more arcs using the same method as the first.
6. Right-click to end the arc function.

You can also enter the data in the **X** and **Y** fields for the coordinating points. Click **Apply**.

Ellipse


The ellipse tool works much the same as the other drawing tools, including three options for construction method. A new method in this tool allows the ellipse to be constructed by defining the angle of the major axis of the ellipse and the height and width. This allows the ellipse to be rotated so that the major axis is not the x axis when it is constructed.

Drawing an Ellipse by Height and Width

Menu: Draw / Ellipse

Toolbar: Construction / Ellipse / Draw by Height and Width



To draw an ellipse by defining a center point and the height and width:

1. Click **Ellipse Icon**. From the Precision Toolbar select **Draw by Height and Width**. 
2. Enter the parameter for the center point.
3. Enter the **Height** and **Width** values.
4. Enter the **Angle** parameter for the ellipse.
5. Click **Apply**. The ellipse will be placed in the workspace.
6. If desired, you can continue to draw more ellipses. Left click to place more ellipses with the same parameters or enter the new values for the ellipses in the fields as before and click **Apply** to draw.
7. Right-click to end the ellipse function.

Drawing and Ellipse by Major and Minor Axis

Menu: Draw / Ellipse

Toolbar: Construction / Ellipse / Major and Minor Axis


1. Click the **Ellipse Icon**. This will open the Precision Toolbar. Select **Ellipse by Major and Minor Axis**. 
2. Left click to place the center point, drag and click to define the angle of the major axis and the width of the ellipse, drag and click to define the height of the ellipse.
3. You can now change to the Ellipse by Height and Width  method to continue placing ellipses with the same parameters. The preview will show on the screen, simply left click to place another ellipse.

Drawing an Ellipse by Corners

Menu: Draw / Ellipse

Toolbar: Construction / Ellipse / Corner to Corner

To draw an ellipse by identifying opposite corners of a bounding rectangle:

1. Click the **Ellipse Icon**. This will open the Precision Toolbar. Select **Draw from Corner to Corner**. 
2. Click to place the first corner, then drag to define the opposite corner. Click the mouse again to complete the ellipse.
3. If desired, you can continue to create more ellipses. Repeat step 2.
4. Right-click to end the ellipse function.




You can also enter the data into the **X** and **Y** fields. Click **Apply**.

Polygon

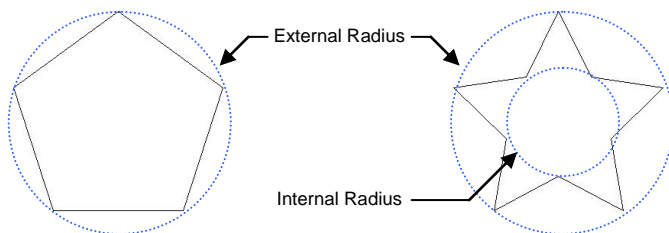
Menu: Draw / Polygon

Toolbar: Construction / Polygon / Convex or Concave

The polygon tool allows you to create multi-sided, closed contours such as pentagons and stars.

1. Click the **Polygon Icon**. This will open the Precision Toolbar. 
2. In the **Precision** Toolbar, select the type of contour you want to create: either a **convex** polygon  or a **concave** polygon. 
3. Set the number of **Points** for your convex shape or concave shape.

- When drawing a convex polygon, simply left click to represent the center of the polygon, drag the mouse to the outside point and then click to end. At this point, you can now enter the coordinates for the center point and one outside point as well as the number of corners for the polygon. Any changes that you make to the coordinates are shown as a preview. Once you click apply the polygon is placed in the workspace.



- If you are creating a concave polygon, set the **Internal Radius** for the circle that the inner vertices intersect.
- Click **Apply** to draw the polygon.
- If desired, you can continue to draw more polygons. Enter the new values for the polygons in the fields as before and click **Apply** to draw.
- Right-click to end the polygon function or press **Close**.

Text

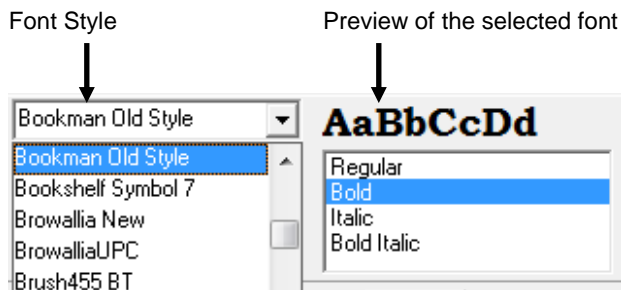
Menu: Draw / Text

Toolbar: Construction / Text

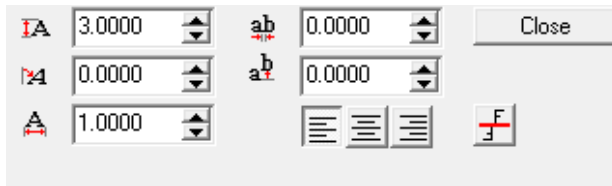
The Text tool allows the user to place text and then edit it to change parameters such as size, font style and spacing. The user is also able to place text on a curve or an arc.

Text Precision Toolbar

Shown below is the first section of the toolbar.



This is the second section of the Precision Toolbar.



Select this option to change the height of the selected text objects. The height of the character is a nominal height not an exact height.



Select this option to change the slant of the characters. A positive value will cause the letter to lean towards the right, while a negative value will make the letter lean back or to the left.



Select this tool to change the width of the selected text objects.



Select this tool to change the spacing between the text objects. A positive value will increase the spacing; a negative value will decrease the spacing between characters.




Select this tool to move the location of the selected text up or down. A positive value will move up; a negative value will move down.



Click on the preferred alignment of the text. Align Left, Align Center, or Align Right.



Select this tool to determine which side of a line or curve the selected text is to be placed.

1. Click the **Add Text Icon**.  This will open the Precision Toolbar.
2. Click to place the cursor in the workspace. **To place the text on an arc or curve**, simply click on the contour at the point you wish to start your text. The icon may show slanted to the angle of the arc or curve. Continue with the following steps.
3. Select the text style. A preview of the text is shown the right of the text selection dialog. All text fonts that are installed in your computer are available to you.
4. You may click the arrows in the height field to adjust the nominal height of the text objects. The cursor will adjust in size to give you a preview of the size.
5. Type the text.
6. Press **Close** to exit the Text tool.



Editing Text Objects

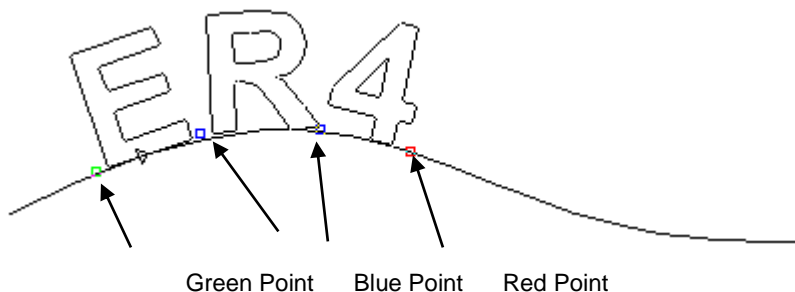
1. Select the text objects to edit by left clicking and dragging the edit selection box around the objects that you wish to edit. Objects can be selected individually or as a group.
2. Make changes to the objects using the tools available to edit the text. A description of the tools is listed in the above section.

Edit Existing Text Object Tool

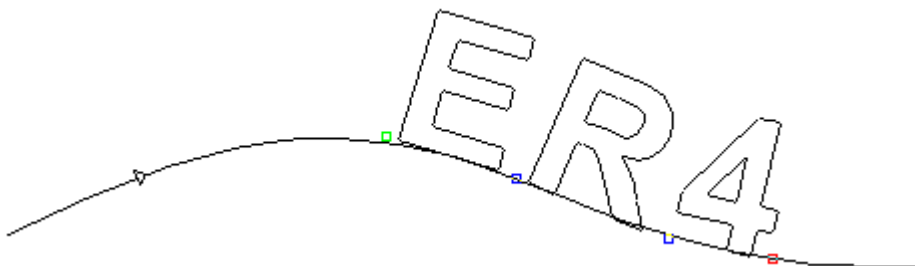
Menu: Draw / Text / Edit Existing Text

Toolbar: Construction / Text / Edit Existing Text

1. Click and hold the **Text Icon**.  This will open the flyout menu. Click on the **Edit Existing Text Objects** icon. 
2. Click on the text to edit.



- Green Point** Click and drag the green point to move the starting point for the line of text objects.
- Blue Point** Click and drag the blue points to adjust the spacing between the letters.
- Red Point** Click and drag the Red point to equally adjust the spacing of the text objects for the entire line of text.



This example shows that the Text was moved using the Green start point.

Convert Text Objects to Curves





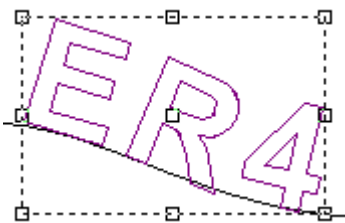
Menu: Draw / Text / Convert Text Objects to Curves

Toolbar: Construction / Text / Convert Text Objects to Curves

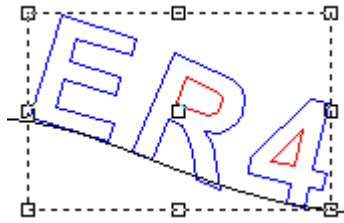
In order to toolpath the Text Objects, they must first be converted to curves.

1. Select the Text Objects.

2. Click and hold the **Text Icon**  to activate the flyout menu. Click on the **Convert Text Objects to Curves Icon**.  The text objects show in a purple color. After the objects have been converted they will show with the standard color for contours.



Text Object



Converted to Curves

Dimensions

Menu: Draw / Dimension

Toolbar: Construction / Dimension

EnRoute contains a variety of dimension tools that you can use to display the measurements of lines and angles. Using the available snap tools along with the dimension tools makes it easy to establish the dimensions of an object.



Single Dimension – This tool allows you to determine the dimension between two selected points.



Angle Dimension – This tool allows you to determine the angle of two selected contours.



Radius Dimension – This tool allows you to determine the radius of a selected circle.



Diameter Dimension – This tool will determine the diameter of a selected contour.




Leader Dimension – This tool allows you to place a leader arrow in the selected location of the drawing.

Single Dimension Tool

Menu: Draw / Dimension / Single Dimension


Toolbar: Construction / Dimension / Single Dimension

1. Click on the **Single Dimension Icon**. 
2. Select the first point.
3. Select the second point.
4. Drag the mouse to the position you want to place the dimension information.
5. Click to place the dimension.
6. You can then go on to create the next dimension or right click to exit the tool.

Angle Dimension Tool

Menu: Draw / Dimension / Angle Dimension


Toolbar: Construction / Dimension / Angle Dimension

1. Click on the **Angle Dimension Icon**. 
2. Select the first contour.
3. Select the second contour.
4. Drag the mouse to the position you want to place the dimension information.
5. Click to place the dimension.
6. You can then go on to create the next dimension or right click to exit the tool.

Radius Dimension Tool

Menu: Draw / Dimension / Radius Dimension


Toolbar: Construction / Dimension / Radius Dimension

1. Click on the Radius Dimension Icon. 
2. Click on the contour.
3. Drag the mouse to the position you want to place the dimension information.
4. Click to place the dimension.
5. You can then go on to create the next dimension or right click to exit the tool.

Diameter Dimension Tool

Menu: Draw / Dimension / Diameter Dimension


Toolbar: Construction / Dimension / Diameter Dimension

1. Click on the **Diameter Dimension Icon**. 
2. Click on the contour.
3. Drag the mouse to the position you want to place the dimension information.
4. Click to place the dimension.
5. You can then go on to create the next dimension or right click to exit the tool.

Leader Dimension Tool

Menu: Draw / Dimension / Leader Dimension

Toolbar: Construction / Dimension / Leader Dimension

1. Click on the **Leader Dimension Icon**. 
2. Click on the contour.
3. Drag the mouse to the position you want to place the Leader lines.
4. Click to place the line. Using the Text tool you are able to place the text on this line if you choose to.
5. You can then go on to create the next dimension or right click to exit the tool.

6. Contour Editing

Editing Points on a Contour

The **Edit Points** tool allows you to manipulate lines, curves and points in contours.

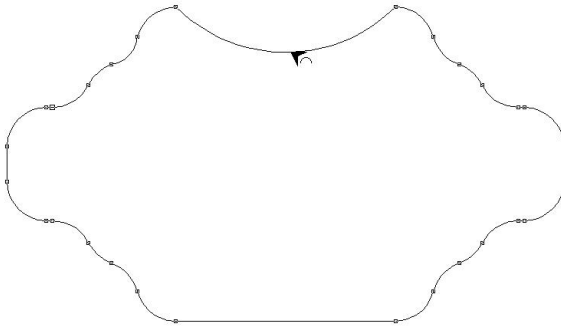
To edit a contour:

1. From the **Draw** menu select **Edit Points**.
2. Select the contour.



Notice that as you place the cursor over a segment, the cursor indicates whether it is a line, arc or curve.

Right-clicking on a segment with the mouse opens a shortcut menu listing commands that are specific to the contour that you are working on.



Exit point edit
Insert point
Delete point(s)
Convert to arcs
Convert to curve
Convert to line
Cut contour
Close contour
Show details


Pressing **F2** will display the Precision Toolbar, which will allow you to precisely place each point.

Editing a Line Segment


1. From the **Draw** menu select **Edit Points**.
2. Select the contour.
3. Click and drag the contour points to move them.
4. To convert a line into an arc, click and drag on the line segment.
5. To convert a line into a curve, right-click on the line and select **Convert to curve**.



Editing a Bezier Curve


1. From the **Draw** menu select **Edit Points**. 
2. Select the contour.
3. Click and drag the contour points to move them.
4. Click and drag on the handles to change the shape.
5. To convert a curve into a line, right-click on the curve and select **Convert to line**.

Editing an Arc


1. From the **Draw** menu select **Edit Points**. 
2. Select the contour.
3. Click and drag the contour points to move them.
4. To change the curve of an arc, click on a point on the arc and drag it in or out.
5. To convert an arc into a line, right-click on the arc and select **Convert to line**.
6. To convert an arc into a curve, right-click on the arc and select **Convert to curve**.

Linearize Curve Section

This tool allows you to edit a section of a contour. It will create a line between two points on the contour, allowing you to eliminate unneeded points quickly.


1. Click and hold the **Edit Points Icon**. This will open the flyout menu, click on the **Linearize Curve Section Icon**. 
2. Click to place the first point on the contour.
3. Click to place the second point on the contour.
4. A pink line will appear to show the preview of the segment to be replaced.
5. Press the **Enter** key to accept the change.

Insert Corner

1. Click and hold the **Edit Points Icon**. This will open the flyout menu, click on the **Insert Corner** icon. 
2. Click to place the first point.
3. Click to place the second point. This will show a preview of the new corner in pink. You can click on the second point and drag it to the desired location. The preview will reflect this change.
4. Press the **Enter** key to accept the change.

Smooth Approximation



1. Click and hold the **Edit Points Icon**. This will open the flyout menu, click on the **Smooth Approximation Tool Icon**. 
2. Click on the contour to place the first point.
3. Click on the contour to place the second point. This will show a pink preview of the new segment. At this point you can adjust the new points and use the handles to change the shape of the new segment.
4. Once you get the new portion of the segment to look the way you want it, press the **Enter** key to accept the change.

Extending Contours

Menu: Draw / Extend

Toolbar: Contour Editing / Extend


The **Extend** tool is used to extend an open contour either interactively, by a specific distance, or to an intersection with another contour segment. The extension is always in a straight line from the end of the contour. If the contour is an arc or curve, the extension is a straight line tangent to the end of the curve or arc.

Extending a Contour Using the Mouse


Menu: Draw / Extend

Toolbar: Contour Editing / Extend / By Graphic

To extend an open contour using the mouse:

1. Click the **Extend Icon**.  This will open the Precision Toolbar. The Method – **By Graphic** -is selected by default.
2. Click on the end of the contour you want to extend.




-  As you move the cursor away from the end of the contour, a preview is provided to show the amount that the contour will be extended.
3. Move the cursor to the new endpoint and click.
 4. After a contour has been extended, you can add another extension to the end of the extension.
 5. Right-click to release the tool or click on the close button.

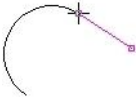
This option can work well with the **Snap** options to extend a contour to a specific point.


Extending a Contour by a Precise Length

Menu: Draw / Extend

Toolbar: Contour Editing / Extend / Precise Length

1. Click the **Extend Lines Icon** ; this will open the Precision Toolbar.
2. In the Precision Toolbar select the **By Length** method and enter the length you want to extend contours by in the length field.
3. Click on the end of a contour to extend it by the specified amount.



 As you move the cursor over the end of an open contour, a preview of the extension is displayed.


4. After a contour has been extended, you can add another extension to the end of the extension.
5. Right-click to release the tool or click on the close button.

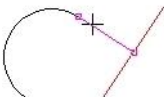
Extending a Contour to Boundaries


Menu: Draw / Extend

Toolbar: Contour Editing / Extend / Boundaries

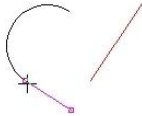
To extend a contour until it intersects with another contour:

1. Click the **Extend Lines Icon** . This will open the Precision Toolbar.
2. Select the **By Boundaries** method.
3. Click on the contours you would like to use as boundaries.
4. To select all contours as boundaries, click on the **All boundaries** button.
5. To clear all selected boundaries, click on the **Clear boundaries** button.
6. Click on the **Extending** button.
7. Click on the end of a contour to extend it to its intersection with the nearest boundary contour.



-  As you move the cursor over the end of an open contour, a preview of the extension is displayed.
8. After a contour has been extended, you can add another extension to the end of the extension. This extension will go out the other side of the boundary, and proceed to the next intersection.
 9. Right-click to release the tool or click on the close button.

If the **Intersection** box is checked, contours can be extended to the intersection with a line tangent to the end of a boundary. If the contour and the boundary are parallel, or if the end of the contour points away from the end of the boundary, no extension will be created. This can be helpful if you need to extend two contours so that they meet at their intersection.



Any open contour in the design can be extended. Even if a contour is selected as a boundary contour, it can still be extended to another boundary contour. This means that you can select all the contours in your design as boundary contours, and then proceed to identify which contours to extend.

Trimming Contours

Menu: Draw / Trim



Toolbar: Contour Editing / Trim

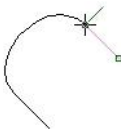
The Trim tool allows you to shorten a contour either interactively, by a specific distance, or based on intersections with boundary contours.


Trimming a Contour Using the Mouse


Menu: Draw / Trim

Toolbar: Contour Editing / Extend/Trim / By Graphic

1. Click and hold the **Extend/Trim Line Icon**  then click the **Trim Contours Icon**.  This will open the Precision Toolbar.
2. Select the **By Graphic** method.
3. Click on the end of the contour you want to trim.



-  As you move the cursor away from the end of the contour, a preview is provided to show the amount that the contour will be trimmed.
4. Move the cursor to the new endpoint and click.

5. Repeat for as many contours as necessary.
 6. Right-click to release the tool or click on the close button.
-  This option works with the snap options to trim a contour to a specific point.


Trimming a Contour by a Precise Length

Menu: Draw / Trim

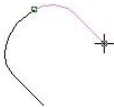
Toolbar: Contour Editing / Extend/Trim / By Length


1. Click and hold the Extend/Trim Line  then click the Trim Contours . This will open the Precision Toolbar.

2. Select **By Length** method and enter the length you want to trim contours by in the field.

 The length that is trimmed is determined by following the path of the contour from the end point, not as a radius or some other method.

3. Click on the end of a contour to shorten it by the specified amount.



 As you move the cursor over the end of a contour, a preview is provided to show the amount that the contour will be trimmed.



4. Repeat for as many contours as necessary.
5. Right-click to release the tool or click on the close button.

Trimming a Contour to Boundaries

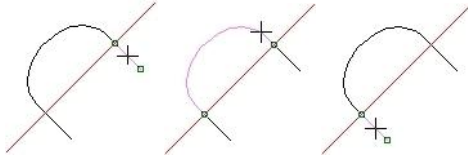
Menu: Draw / Trim


Toolbar: Contour Editing / Extend/Trim / By Boundaries

To trim a contour at the points where it intersects with another contour:

1. Click and hold the **Extend/Trim Lines**  then click on the **Trim Contours** .
2. In the Precision Toolbar, click on the **By Boundaries** method.
3. Click on the contours you would like to use as boundaries.
4. To select all contours as boundaries, click on the **All boundaries** button.
5. To clear all selected boundaries, click on the **Clear boundaries** button.

6. Click on the **Trimming** button.
7. Move cursor over the contour to be trimmed. A preview will highlight the part of the contour that will be trimmed. Left click on the mouse to activate the trim.



 As you move the cursor over the end of an open contour, a preview of the trimmed area is displayed.

8. Repeat for as many contours as necessary.
9. Right-click to release the tool or click on the close button.

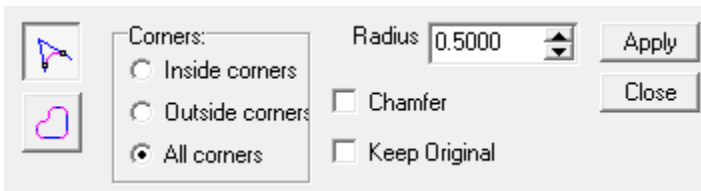
Filleting Contours

Menu: Draw / Fillet

Toolbar: Contour Editing / Fillet

The fillet tool inserts an arc of a defined radius at selected corners of one or more contours.

Fillet Precision Toolbar



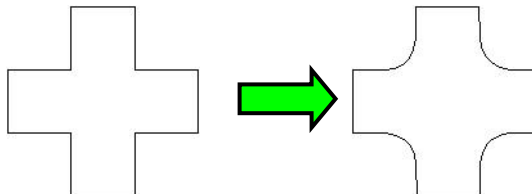
Single Fillet

The single fillet allows you to select one specific corner to fillet.

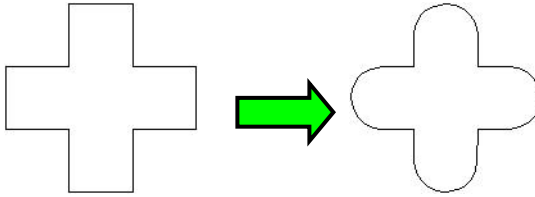
Multiple Fillet

This allows you to choose to Fillet All corners, Outside corners or Inside corners all at the same time with the same parameter.

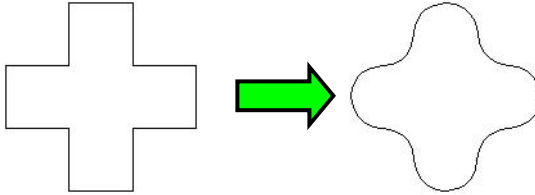
Inside Corners



Outside Corners



All Corners



Radius

This is the parameter for the size of the radius for the Fillet or Chamfer.

Chamfer

Select this box if you would like to chamfer the corner instead of fillet.



Keep Original

When this box is selected, the original will be kept and an open contour segment will be placed where the fillet was previewed.

Single Fillet

Menu: Draw / Fillet




Toolbar: Contour Editing / Fillet / Single Fillet

1. Click the **Fillet Icon**.  This will open the Precision Toolbar.
2. Click the **Single Fillet Icon**. 
3. Select the first segment of the contour.
4. Select the second segment of the contour.
5. Enter in the radius dimension. This will show a preview of the fillet.
6. Click **Apply**.

Multiple Fillet

Menu: Draw / Fillet

Toolbar: Contour Editing / Fillet / Multiple Fillet


1. Click the **Fillet Icon**.  This will open the Precision Toolbar.
 2. Click the Multiple Fillet **Icon**. 
 3. Select the contour.
 4. Choose one of the Corner parameters: **All** corners, **Inside** corners, or **Outside** corners.
 5. Enter in the radius dimension. This will show a preview of the fillet.
 6. Click **Apply**.
-  Notice that where the fillets overlap, the software automatically makes them tangent with each other.

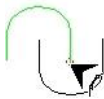
Joining Contours


Menu: Draw / Join

Toolbar: Contour Editing / Join

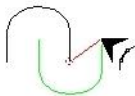
The Join tool allows you to join open contours to create larger contours.


1. Click on the **Join Icon**. 
2. Click on the end point of one of the contours you wish to connect.



 When you put the cursor over the end of a contour, the contour is highlighted.

3. Click on the end point of the second contour that you wish to connect.



 You will notice that EnRoute draws a red preview line between the two contours and highlights the second contour.




The joined contours are now one large contour.

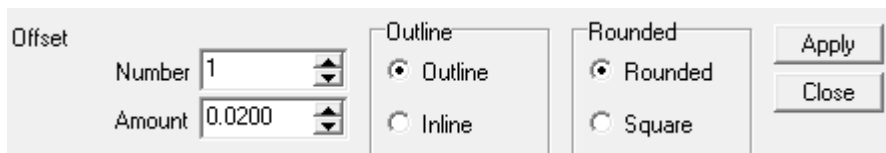
Offset Contours

Menu: Draw / Offset Contours

Toolbar: Contour Editing / Offset Contours

This tool is used to create a contour that is offset from the selected contour.

1. Select the contour.
2. Click the **Offset** icon.  This will open the Precision Toolbar.



The Precision Toolbar for Offset Contours contains the following elements:

- Offset** section: Two numeric input fields. The first is labeled "Number" and contains the value "1". The second is labeled "Amount" and contains the value "0.0200". Both fields have up and down arrow buttons.
- Outline** section: Two radio button options. The first is labeled "Outline" and is selected. The second is labeled "Inline" and is unselected.
- Rounded** section: Two radio button options. The first is labeled "Rounded" and is selected. The second is labeled "Square" and is unselected.
- Buttons**: Two buttons on the right side, labeled "Apply" and "Close".


3. Enter the number of Offset contours desired.
4. Enter the amount of the offset.
5. Select Outline or Inline.
6. Select Rounded or Square.
7. A preview of the Offset is shown on the screen.
8. Click **Apply** to accept.

Partial Offset Contours

Menu: Draw / Offset

Toolbar: Contour Editing / Offset Contours / Partial Offset



This tool is used to create a contour that is offset from the selected contour.

1. Select the contour.
2. Click and hold the **Offset** icon then click the **Partial Offset Icon**.  This will open the Precision Toolbar.



The Precision Toolbar for Partial Offset Contours contains the following elements:

- Partial offset** section: A checkbox labeled "Partial offset" which is checked.
- Offset** section: A numeric input field labeled "Offset" containing the value "0.5000" with up and down arrow buttons.
- Buttons**: A button labeled "Close" on the right side.

3. Select the square  or the rounded  offset icon.
4. Click on the segment or segments that you wish to offset. Hold the shift key to select more than one segment at a time.
5. Enter the amount of the offset.

6. Click the mouse on the side of the segment that you want the offset to be placed.
7. Press Close to exit the tool.

Merging Selected Open Contours

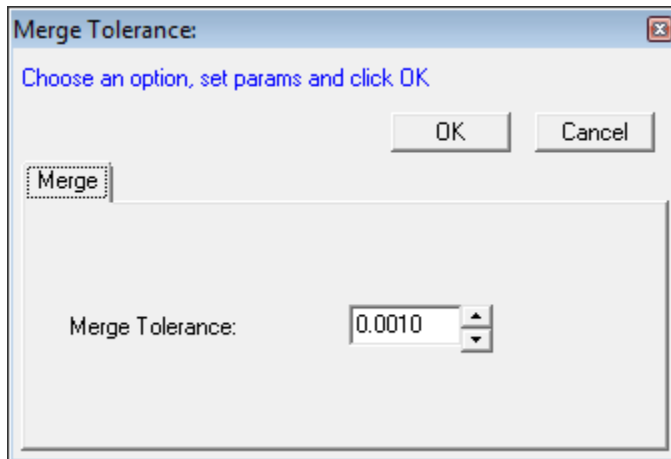
Menu: Transform / Merge Selection

The Merge Selection tool is used to join open contours together based on a definable tolerance.

- ☰ This capability can be very useful if you are using contour information that has been generated in a different software package. Many CAD programs are oriented toward drafting functions, and don't place an emphasis on closed contours as is necessary in order to create toolpaths. After importing a file into EnRoute, this tool provides a means of quickly joining separate contours into closed shapes that are more compatible with toolpath creation.

To join two or more open contours:

1. Select the open contours.
2. From the **Transform** menu, select **Merge selection**.




3. Define the **Merge Tolerance** (the maximum distance endpoints can be separated by and still be connected).
4. Click **OK**.

Explode Selected Contours

Menu: Transform / Explode Selected Contours

This tool is used to separate a contour into individual segments.


To explode a contour:

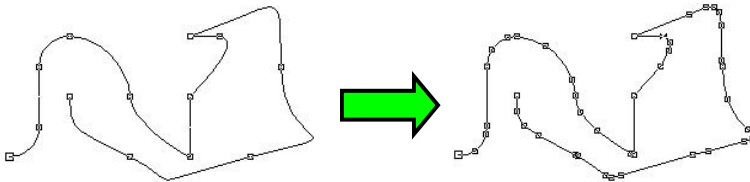
1. Select the contour.
2. From the Transform menu, select **Explode Selected Contours**. 
3. This will create individual segments for the contour.

Converting Contours to Arcs


Menu: Transform / Convert Curves to Arcs

The Convert Selection to Arcs tool transforms the contours in the selection, converting the lines and Bezier curves into arcs.

 When a curve is converted to arcs, one curve may require several arcs, so the number of segments may increase significantly.



To convert contours into arcs:

1. Select the contours.
2. From the **Transform** menu, select **Convert Selection to Arcs**. 


Reverse Open Contours

Menu: Transform / Reverse open Contours

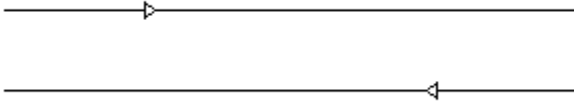
This tool reverses the direction of an open contour.

The direction of the contour is indicated by the direction of the arrow head on the contour. To display this option, go to the **Setup Menu, Preferences, View Setup** and check the box for **Open contour** direction.

To reverse the direction of the contour:

1. Select the contour.
2. From the Transform menu select Reverse open contour. 

The example below shows that the arrow of the first line segment was pointed to the right. After it was selected and reversed, the arrow is showing to direction to the left.



Creating Multiple Copies of a Contour

Array Copy

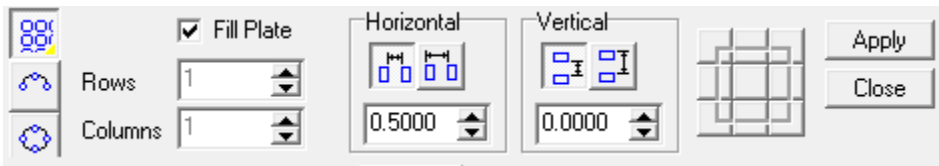
Menu: Draw / Multicopy


Toolbar: Contour Editing / Multicopy


The Array Copy tool produces multiple copies of the selected contour, ordered into a specified number of rows and columns with a specified spacing.

For example, if you need to cut out several copies of the same contour, it is a simple task to create those copies at the desired spacing on the sheet using this tool.


1. Click on the **Multicopy Icon**.  This will open the Precision Toolbar.

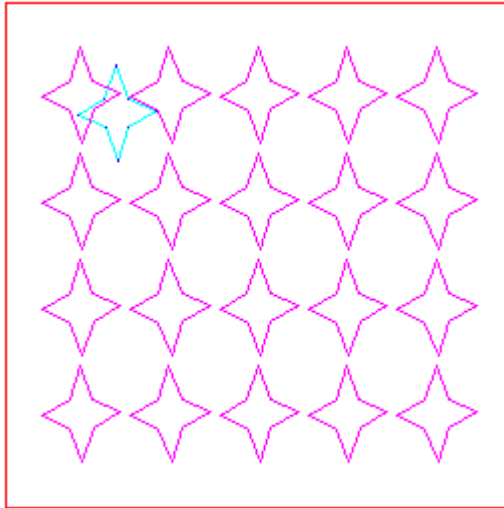


2. Select **Array Copy**. 
3. In the Precision Toolbar, enter the number of rows and columns of copies desired, and the space between the rows and columns. A preview of the copies will be shown in pink.

 If you have one or more contours that must be copied so they fill up your whole piece of material. (For example a 4 x 8 sheet): Simply check the **Fill Plate** Box. The selected contours will be copied to fill the entire plate. You do not need to fill in the number of horizontal or vertical copies. The spacing between the contours is based on the spacing between the rows and columns that have been defined.

4. Click **Apply** to place the copies.

 This is an example of an array copy to fill the plate, with .10 horizontal spacing and .10 vertical spacing.





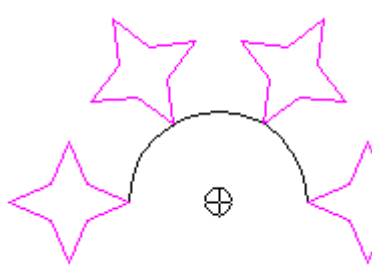
Arc Copy

Menu: Draw / Multicopy

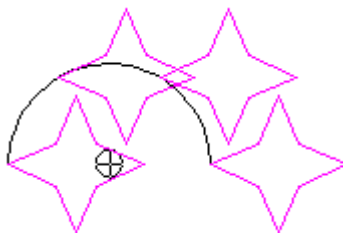
Toolbar: Contour Editing / Multicopy

The **Arc Copy** tool takes an existing contour or set of contours and creates a series of duplicates spaced out along the curve of an arc. The arc of copies that is created is defined by the radius and angle of the arc, the number of copies that are spaced along that angle, and the positioning point that will be used to position the copies on the arc.

1. Click the **Multicopy Icon**.  This will open the Precision Toolbar.
2. Select **Arc Copy**. 
3. Select the object to copy. A preview will show on the screen as you enter information into each of the parameters in the Precision Toolbar once you have selected the object to copy.
4. Enter the **Number** of copies that will be spaced along the arc.
5. Enter the **Radius** of the arc.
6. Enter the **Start Angle** (in degrees). This represents the point at which the arc begins.
7. Enter the **Swept Angle**, which defines the portion of a circle (in degrees) that will be used by the arc. If more than 360 degrees are entered, the arc will loop around.
8. Check **Align to Path** to rotate the copies along the curve of the arc.



Aligned



Not Aligned

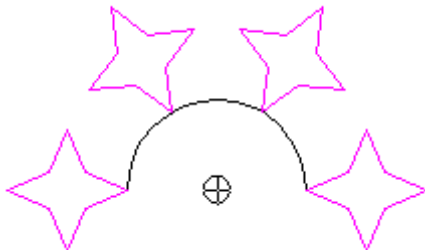
9. Click the reference grid button that corresponds to the corner, side or center point that will be used to position the contours along the arc.

These are the parameters used in the below example.

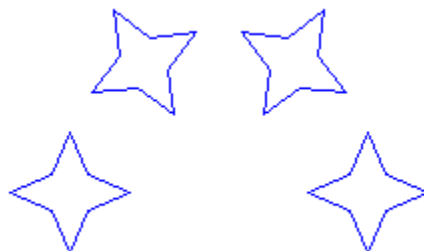
	<input checked="" type="checkbox"/> Align	Number	4	Start Angle	0.0000	Center X	2.5000		Apply
	Radius	1.0000	Swept Angle	180.0000	Y	1.0000	Close		
				Z	0.0000				



**Object with
positioning points**

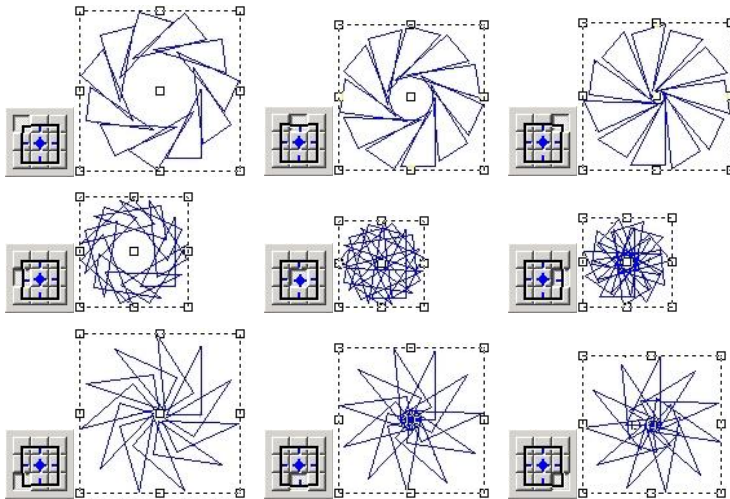


Preview of Arc Copy



Result of Arc Copy

The positioning point that is chosen can have a major effect on the appearance of the final set of copies. The examples below are all copies of the same right triangle, and use identical parameters apart from the positioning point selected.





Path Copy

Menu: Draw / Multicopy


Toolbar: Contour Editing / Multicopy

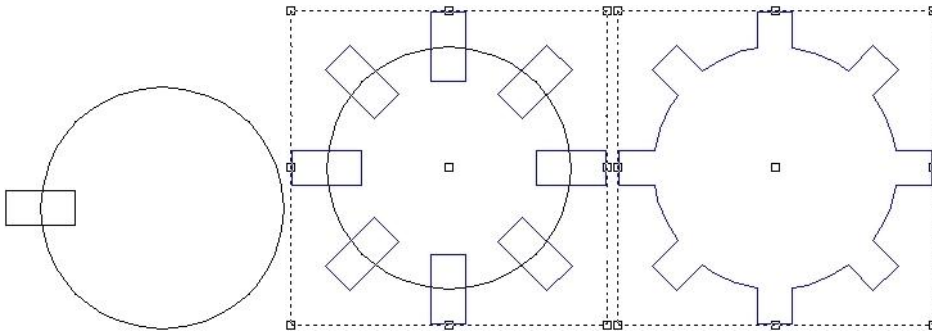
The Path Copy tool allows a shape to be copied along another contour by defining the number of copies, spacing, and how the contours are to be aligned.

1. Click **Multicopy**.  This will open the Precision Toolbar.
2. Select **Path Copy**. 
3. Select the object to copy. Then select a contour for the path by clicking on it. A preview will show on the screen as you enter information into each of the parameters in the Precision Toolbar.
4. Enter the **Number** of copies. The spacing will be adjusted accordingly.
5. Enter the **Radius** of the arc.
6. Enter the **Offset Percentage**. Use Offset if you want the original contour offset from the starting point of the path contour. The percentage is based on the spacing.
7. Check **Align to Path** to rotate the copies to match the direction of the path.
8. Select one of the following copy placement options:

Hold Number	Enter the number of copies and the spacing will be calculated to keep them evenly spaced.
Hold Spacing	Enter the desired spacing and the program will calculate how many copies will fit in the path.
All Corners	Copies will be placed at both inside and outside corners along the path.
Inside Corners	Copies will be placed only at inside corners on the path.
Outside Corners	Copies will be placed only at the outside corners on the path.

9. Click the reference grid button that corresponds to the corner, side or center point that will be used to position the contours along the arc.
10. Click the reference grid button that corresponds to the corner, side or center point that will be used to position the contours along the arc.
11. Click **Apply**.

 The example below shows how to create a gear shape by first drawing a circle and a rectangle, and then copying the rectangle around the circle. The Weld Joined command is used to combine the shapes.



Vectorizing Bitmaps

Menu: Draw / Vectorize Bitmap

Toolbar: Contour Editing / Vectorize Bitmap

Since the software uses vector-based contours to create toolpaths, any bitmap artwork you may want to use in your design must first be converted into a vector form. This technique is called vectorization.

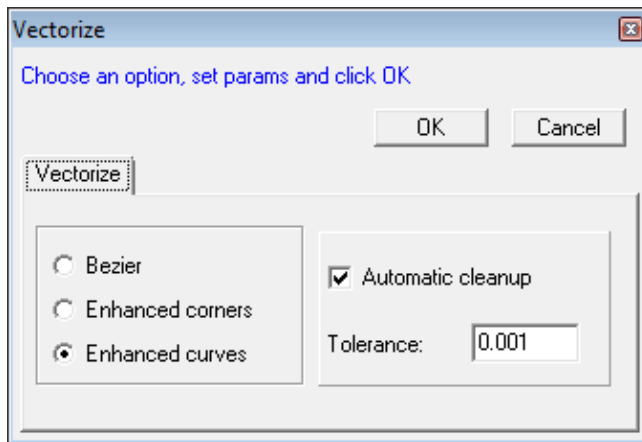
The Vectorize Bitmap tool allows you to trace bitmap art and convert it into contours.

Before vectorizing a bitmap, you should clean up the bitmap as much as possible using a bitmap editor. Remove specking and unwanted detail. Make sure that contours you want to turn into closed contours are completely closed in the bitmap. If possible, convert the image to a grayscale or black and white image.

1. Import the bitmap into your design and select it.

For information on importing a bitmap or other graphic, see “Importing a Design”.

2. Select **Vectorize bitmap**.



In the Vectorize dialog, select the **Bezier**, **Enhanced corners**, or **Enhanced curves** option.

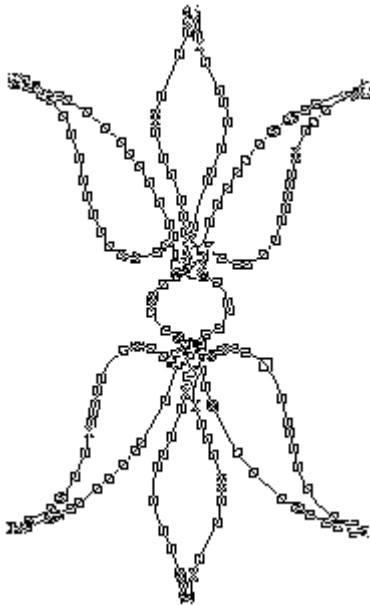
- The **Bezier** option emphasizes using Bezier curves as it completes the vectorization process.
 - The **Enhanced Corners** option emphasizes creating a contour with distinct sharp corners. It also provides a number of additional parameters that affect the vectorization process.
 - The **Enhanced Curves** option emphasizes using arcs to provide a best fit of the vectors to the original artwork.
3. Check the **Auto cleanup** box if you want the software to attempt to automatically simplify the contours it creates. Set **Tolerance** to the size of the smallest features on the bitmap that you want the software to keep. See “Cleaning Up Contours” for details.

Cleaning Up Contours

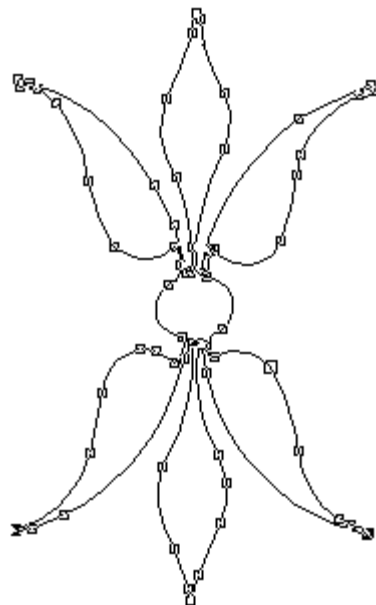
Menu: Draw / Cleanup

Toolbar: Contour Editing / Cleanup selected contours


The Cleanup tool automatically simplifies the selected contours so that they use fewer segments to achieve the same effect. This makes it easier for the software to create toolpaths, and reduces the chances for the toolpath creation process to fail because of some odd configuration in the geometry.



304 Segments



65 Segments

1. Select the contour.
2. Select **Cleanup**. 
3. Set **Tolerance** to the size in inches of the smallest features you want the software to keep.
4. Higher **Tolerance** settings will cause the contour to be simplified more, but they will also change the appearance of the contour more.
5. Click **OK**.

7. Arranging Contours

Selecting Contours

To select a contour, click on its edge.

You can also select multiple contours using one of the following methods:

- Hold down the **SHIFT** key and click on the contours one after the other.
- Click and drag to draw a selection box around the contours.
- From the **Edit** menu, select **Select All** to select all contours in the current design.

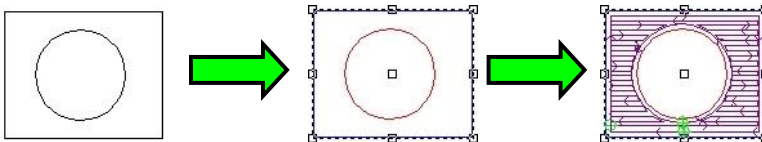
Automatic Sorting for Open Curves, Containers and Holes

Whenever one or more contours are selected, the software automatically checks to see whether the contours are open curves, containers or holes.

By default, all closed curves are classified as containers.

If a closed contour is completely contained within another closed contour, the outside contour is identified as a container, and the inside contour is identified as a hole in that container. The contour of the hole forms an inner edge to the container contour.

The following illustration shows two contours being selected, converted to a container and hole, and being filled with a Hatch Fill toolpath.



By default, open curves are color-coded purple, containers are color-coded blue, and holes are color-coded red.

Cutting, Copying and Pasting Contours

Menu: Edit / Cut or Paste

Toolbar: File / Cut or Paste

To cut and paste contours:

1. Select the contours to cut.
2. From the **Edit** menu choose **Cut** or press **Ctrl+X**.
3. Select the design where the contours are to be pasted.



4. From the **Edit** menu choose **Paste** or press **Ctrl+V**.



When there is more than one design window open, use the Cut and Paste commands to remove a group from one design and place it in another. You can continue to paste the cut selection until something else is placed onto the Clipboard.

Copying and Pasting

Menu: Edit / Copy or Paste

Toolbar: File / Copy or Paste

1. Select the contours to copy.

2. From the **Edit** menu select **Copy** or press **Ctrl+C**.



3. If the copied image is to be pasted into a different design, select the design where the contours are to be pasted.

4. From the **Edit** menu select **Paste** or press **Ctrl+V**.



When pasting into the same design, the copied image will be placed in the same place as the original. Move the original image to the new location before pasting.

If you want to make a copy within the same design, press the CTRL key, click on the center control box and drag a copy to a new location.

Pasting from Other Programs

See the Preference command located in the Setup menu for options regarding Merge Contours and Automatic Cleanup.

Deleting Contours

Menu: Edit / Delete

Toolbar: File / Delete

1. Select the contours.

2. From the **Edit** menu, select **Delete**.



You may also select the contours to be deleted and press the **Delete** key instead of selecting the **Delete** command.


Grouping and Ungrouping Contours

Menu: Transform / Group or Ungroup

Toolbar: Object Edit / Group or Ungroup


Grouping contours together allows you to select and manipulate a group of contours as a single contour. After grouping contours together, you can also Ungroup them to return them to their original state. This tool is used mainly as a convenience when you are working on intricate designs.

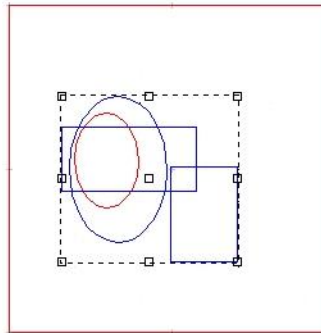
1. After creating a number of contours, select them.

 Notice that the contours are automatically sorted when they are selected.

2. Click on the **Group Icon**.




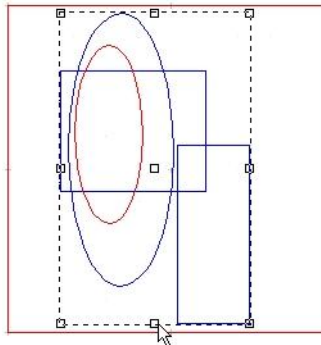
 After selecting the contours you want to group you can press **Ctrl+G** to group them.



Now, when you change the size of the group, or move it around, it will be treated as a single object.

If you select any part of the group to work with, you will be selecting the entire group. You can use the control points, or handles, to scale the entire group as a unit.

 To separate the group into its component contours, simply select the group and click on the **Ungroup Icon**, or press **Ctrl+U**.



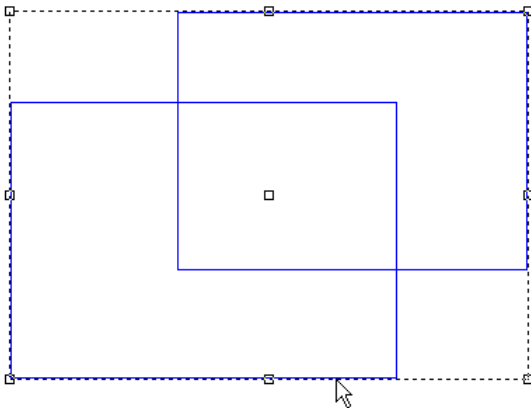
Moving, Scaling, and Rotating Objects

EnRoute provides several methods for interactively and precisely moving, scaling and rotating objects. It is important to be able to quickly move and scale objects in the drawing, but it is equally important to be able to position and size objects very precisely. EnRoute's tools meet both of these needs.

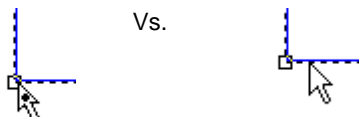
Moving and Scaling Interactively

Within EnRoute's standard interface it is possible to move and scale objects simply by selecting them and clicking and dragging. For creative design work that doesn't necessarily require precise placement or sizing, this is the easiest and quickest method.

In order to move a selection, it is as easy as clicking on one of the contours in the selection and then dragging the selection to a new location. It is important, though, to click on one of the selected contours in order to maintain the selection. The image below illustrates this – note that the cursor is on top of one of the contours in the selection.

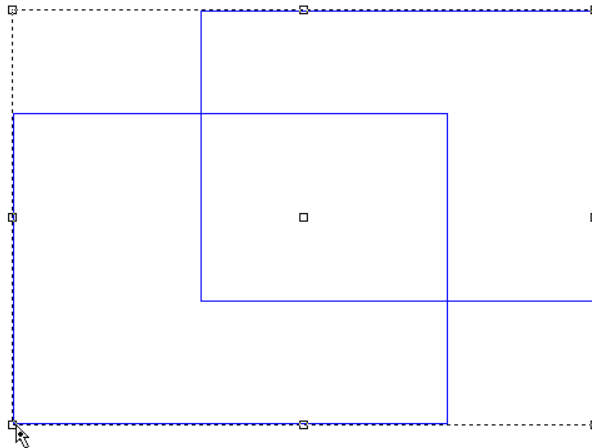


It is also possible to move a selection using one of the nine selection handles. The easiest way is to place the cursor on the center handle and then click and drag the selection to a new location. It is also possible to move a selection using one of the other handles but this takes a little practice. If you place the cursor directly on top of one of the handles, notice that the center of the cursor arrow turns black. If you click and drag at this point, you will scale the selection rather than move it.



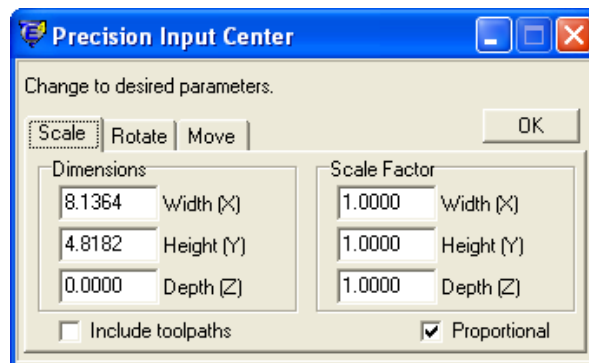
If, instead of placing the cursor directly on top of the handle you place the cursor near the handle and then click and drag, the selected point for the move will automatically snap to the corner of the selection and then allow you to move the selection to the desired location. After you master the concept of clicking near a corner this is a handy way to precisely position a selection.

Scaling a selection interactively is accomplished by placing the cursor on one of the selection handles and then clicking and dragging to move the handle and scale the selection. If you also press the Shift key while dragging the handle the selection will be scaled proportionately.



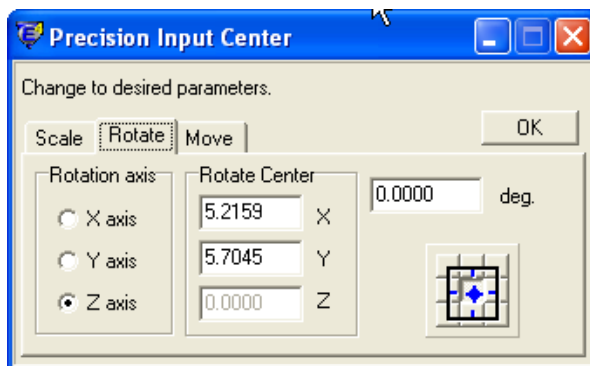
Moving, Scaling and Rotating using the Precision Input Center

The **Precision Input Center** has been used in EnRoute historically as the method for precisely locating objects. Although new tools are now available for accomplishing these tasks, the Precision Input Center is still available for this purpose. One of its advantages is that it provides a simple method for precisely sizing and locating objects.

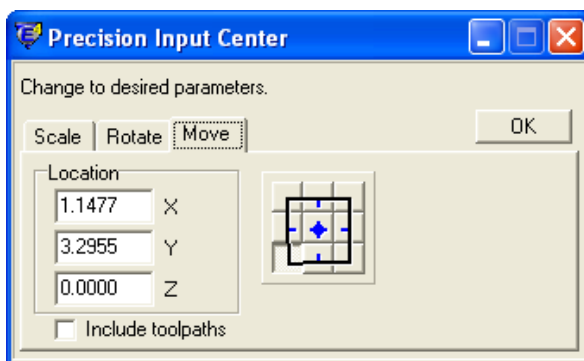


The Precision Input Center is activated by pressing the F2 key. The selection can be scaled by entering values either for Dimensions or Scale Factor. In order to proportionally scale the selection, make sure that the Proportional checkbox is checked. Click on the OK button in order to complete the scaling activity.

Rotating the selection using the Precision Input Center is similar to scaling. After selecting the objects to be rotated and activating the Precision Input Center, click on the Rotate tab. The rotation axis will be based on the active viewport. From the top view you will typically rotate the selection about the Z axis. This means that the objects are rotated in the X-Y plane. Enter the desired rotation angle and click on the OK button to complete the rotation. The center of rotation defaults to the center of the selection; however, it can be placed to a different location if desired.



In order to Move the selection, click on the Move tab, select the corner of the selection for which you want to specify the location, and then enter the coordinates for that location. Click on the OK button to execute the move.

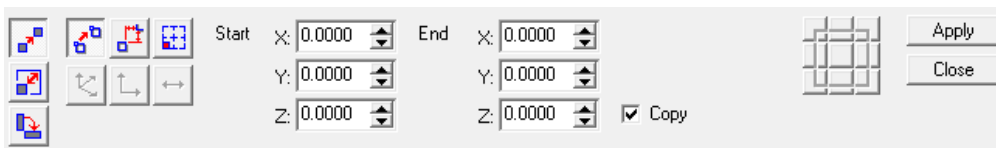


When specifying the corner of the selection to move, the active viewport defines which plane is active. In the top view, the X-Y plane is active and the corner points are located on the plane nearest the viewer. For reliefs, this means that the points are located on the top of the relief. In the front view, the corner points are located on the X-Z plane on the side of the object closest to the viewer.

Note: The Include Toolpaths checkbox lets you choose whether toolpaths are included in the size and location coordinates for the selection.

Move, Scale, Rotate Tools

EnRoute also includes tools for moving, scaling, and rotating objects that are quite flexible and that provide a number of options for accomplishing these edits. These tools are accessed from the Object Edit toolbar and the Transform menu item. Additionally, you can move between these functions from within the Move-Scale-Rotate toolbar after it has been activated. A graphic of this toolbar is shown below.






The following sections provide a detailed description of the parameters for each of the tools and also provide some step-by-step direction.

Move Tool

Menu: Transform / Move


Toolbar: Object Edit / Move

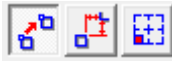
The Move tool provides three different methods for moving selected objects. Within each of these methods you have the option of either entering coordinate information to define locations or defining locations interactively in the drawing.

Move Absolute 	The Move Absolute method allows you to define a starting point and ending point for the move. Objects can be positioned very precisely by defining locations for the move based on entered coordinates. With the point snap options it is also an easy task to position the selection precisely by clicking in the drawing.
Move Relative 	This method is similar to the absolute method, with the exception that it allows you to define the amount of the move in each direction. Use it to move the selection over 5 and up 3.
Move Corner 	Use this method to position a selected corner to a specific point. It is an easy way to move the lower-left corner of the selection to a specific point.
Start	In the Absolute mode this is the coordinate of the start of the move operation.
End	In the Absolute mode this is the coordinate of the end of the move operation.
Origin	In the Relative mode this is the coordinate of the start of the move operation.
Move	In the Relative mode this is the amount of the move in each direction.
Copy	By checking this checkbox you can create a copy of the selection when it is moved. The original will stay in place.
Corner Selector	In the Corner mode this lets you choose which corner of the selection for which to specify a new location.

To Move a selection:

1. Select the objects to be moved.

2. Click on the Move button,  select it from the Transform menu or press Shift+F5. This will activate the Precision toolbar for the Move tool.

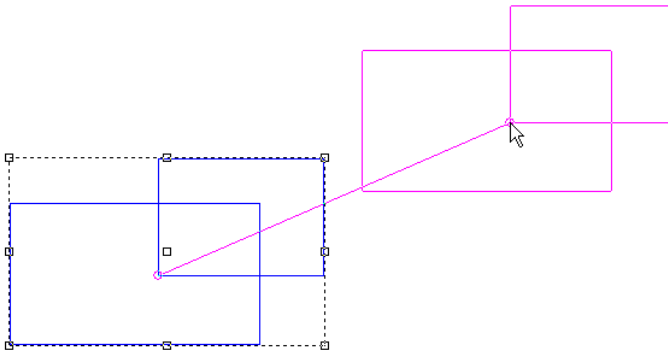


- Choose the desired move Mode. In this case choose Move



Absolute.

- Define the coordinate of the start point of the move. This can be done by typing in the coordinate values or by interactively clicking on a start point.
- Define the end point of the move by typing in the end point coordinate or by clicking on the end point in the drawing.
- If you click on the end point then the move will be completed. If you define the coordinate of the end point then click on the **Apply** button to complete the move.



Key Concept with Right Click – The right-click button on the mouse can be used to ‘back out’ of the Move-Scale-Rotate tools. When you right click while you are in the tool, the effect will be to back you up one step. For example, in the image above the user has selected the objects to be moved, clicked on the start point for the move and is in the process of locating the point for the end point of the move. A right click at this point would back the user to the point of defining the start point of the move. Another right click would deselect the two rectangles. Another right click would exit the move tool.




Scale Tool








Menu: Transform / Scale

Toolbar: Object Edit / Scale

The Scale tool provides several options for scaling selected objects. For scaling interactively the scale tool uses a reference line to define the base point of the scale and the scale factor. The tool also allows you to enter a precise size or scale factor for the scaling operation.


Scale Three Dimensions		When the selection is scaled, it will be scaled in all three dimensions regardless of what view is active. This applies mostly to three-dimensional objects such as Reliefs and Meshes.
Scale Two Dimensions		When the selection is scaled, only the two visible dimensions will be scaled. In the Top View only the X and Y dimensions will be scaled. This is helpful when scaling a Relief when you don't want the thickness of the relief to change.
Scale One Dimension		When the selection is scaled interactively using the reference line, it will be scaled along the direction of the reference line. This will have the effect of distorting the scaled objects in the direction of the reference line. If the reference line is defined either horizontal or vertical then the scaling will be along the X or Y axis. This option will likely be used less often than the other two scaling options as more of a specialty tool.
Size		Precisely scale the selection by entering the desired XYZ size.
Factor		Precisely scale the selection by entering the scale factor.
Include Toolpaths		Include any toolpaths in the selection in the Size parameter.
Proportional		If checked, any scaling operation will be performed proportionally to the selection.
Copy		By checking this checkbox you can create a copy of the selection when it is scaled. The original will stay in place.
Corner Selector		This selector is used when Size or Factor parameters are used to scale the selection.



To Scale a selection precisely:

1. Select the objects to be scaled.
2. Click on the Scale button,  select it from the Transform menu, or press Shift+F6.
3. Choose the scaling option you prefer.    In this case, choose Scale Three Dimensions. 
4. If you want the scale operation to be proportional, make sure this checkbox is checked.
5. If you want to make a copy of the selection, make sure this checkbox is checked.
6. Enter either the desired Size or Scale Factor.
7. Choose the desired corner to serve as the anchor point for the scaling operation.
8. Click on the **Apply** button to perform the scaling operation.

To Scale a selection interactively:

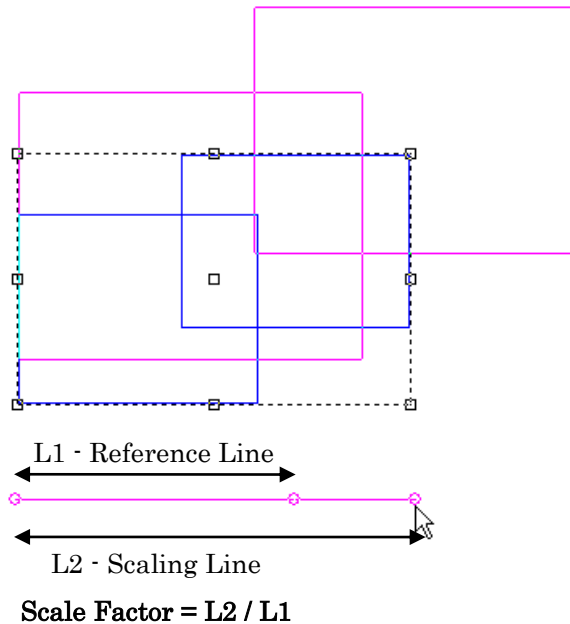
1. Select the objects to be scaled.


2. Click on the Scale button,  select it from the Transform menu, or press Shift+F6.

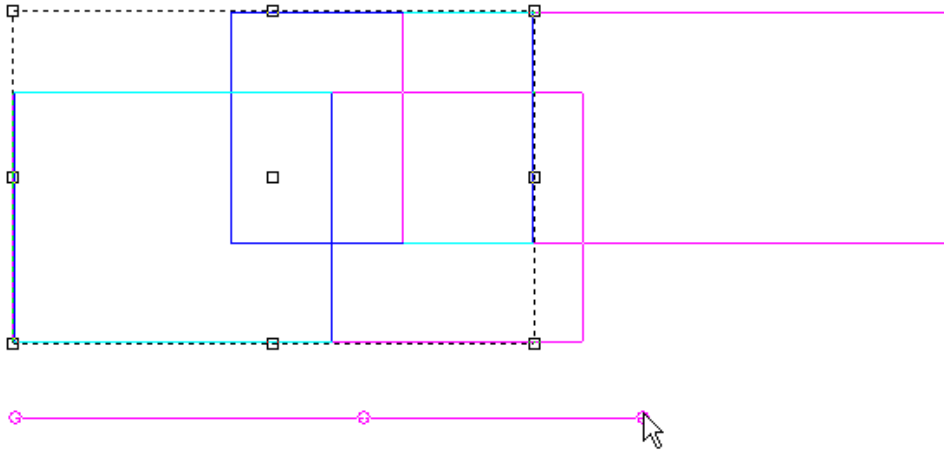
3. Choose the scaling option you prefer.  In this case choose Scale Two Dimensions. 

4. Click on the point you would like as the anchor point for the scaling operation. This can be a specific point if you would like the selection to remain in a specific location, or it can be any point on the screen if the location is not important. This point is the start point of the reference line.
5. Click on another point that will serve as the end point of the reference line. Again, this point can be a specific point so that the reference line is a specific length, or it can be any point in the view.
6. The next point defines the length of the scaling line. The scale factor for the scaling operation is based on the proportional length of the reference line relative to the scaling line. Both lengths are defined from the anchor point. After the scaling line is defined then the scaling operation is completed.

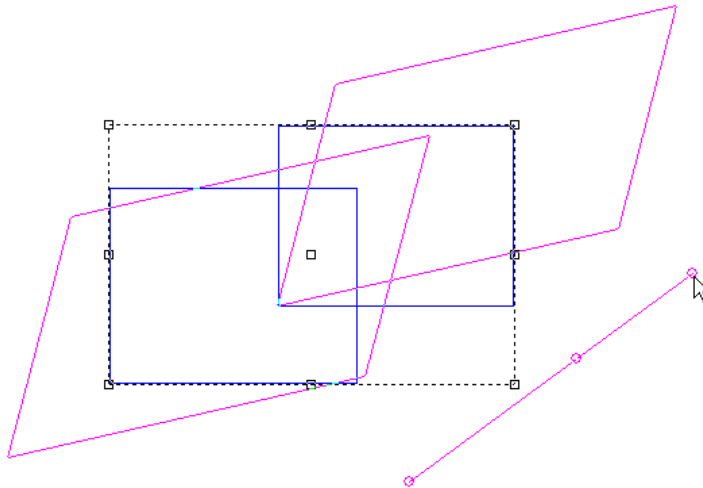
The following graphic illustrates how the Reference Line and Scaling Line lengths are used to determine the Scale Factor.



Using the Scale One Dimension option  will scale the selected objects along the reference line. If the reference line is drawn horizontal, then the scaling will be done along the X axis. If the reference line is drawn vertical, then the scaling will be done along the Y axis. If the reference line is drawn at some other angle, then the selection will be distorted along the line. The following images show one-dimensional scaling along the X axis and along another line.



Scaling along the X Axis



Scaling along an arbitrary line

Rotate Tool

Menu: Transform / Rotate


Toolbar: Object Edit / Rotate

The Rotate tool allows you to rotate selected objects both precisely using a defined rotation center point and a rotation angle, and interactively by defining a rotation center, a starting direction, and a finished direction. You can also choose to perform the rotation using a combination of interactive and precise input.


Following is a definition of the elements of the Rotate toolbar:

Center	This allows the rotation center to be defined precisely.
Axis	This is the axis about which the rotation will be completed. In the top view the axis will most often be defined as the Z axis.
Angle	This is the rotation angle.
Copy	If this is checked then the rotated objects will be a copy of the originals.
Corner Selector	This defines the location of the rotation center when rotating the selection using the Angle parameter in the toolbar.

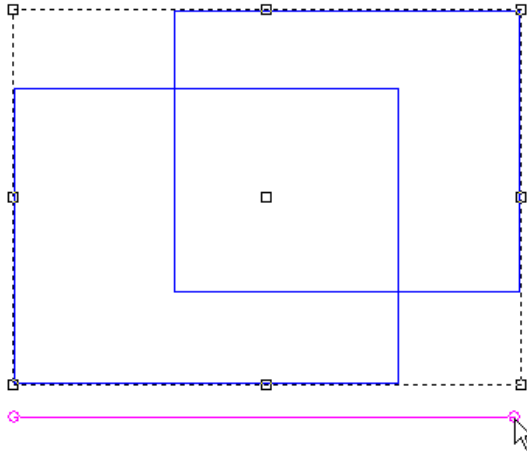
To Rotate a selection precisely:

1. Select the objects to be rotated.
2. Click on the Rotate button,  select it from the Transform menu, or press Shift+F7.
3. Enter values for the Center of rotation or select the desired corner from the Corner Selector.
4. If you want to make a copy of the selection, make sure this checkbox is checked.
5. Enter the Angle.
6. Click on the **Apply** button to perform the rotation.

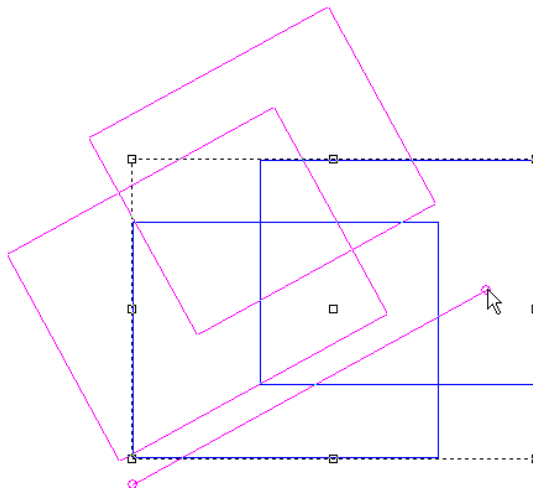
To Rotate a selection interactively:

1. Select the objects to be rotated.
2. Click on the Rotate button,  select it from the Transform menu, or press Shift+F7.
3. Click on a point to define it as the Center of rotation.

- Click on another point to define the initial direction. The line between the first clicked point and the second clicked point defines the “zero” direction for the rotation. As you move the mouse around you will see a preview of the rotation.
- Click on a point to define the angle of rotation and to complete the rotation.



The first click defines the center of rotation and then the second click defines the reference line for the rotation.




Moving the mouse provides a preview of the rotation. The third click defines the angle and completes the rotation.

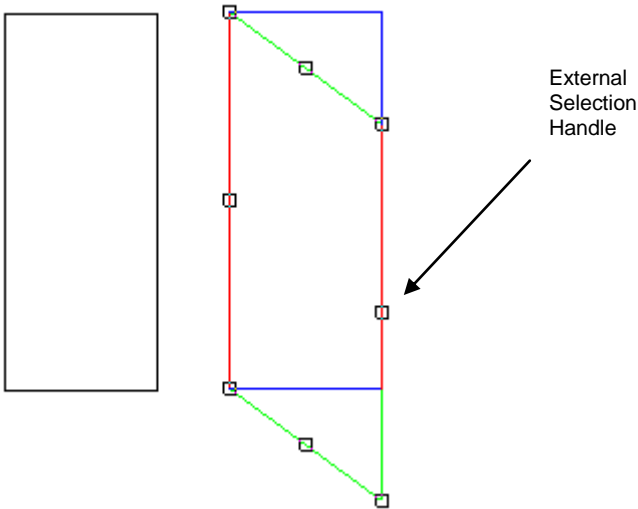
Distort Contours

Menu: Transform / Scale

Toolbar: Object Edit / Scale / Distort

The Distort tool allows you to stretch an object disproportionately.

1. Select the contour.
2. Click and hold the **Scale Icon** then click the **Distort Tool**. 
3. Click on one of the external selection handles and drag the mouse to the desired position.
4. Press **Enter** to accept the change.
5. **Right Click** to end the tool.




Mirroring a Contour

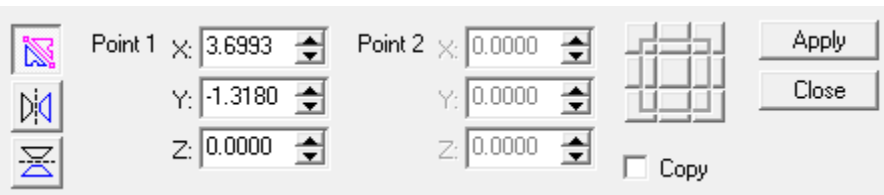
Menu: Transform / Mirror

Toolbar: Object Edit / Mirror

This tool reverses the selected contour so that it becomes its mirror image. This can be done interactively by defining the axis around which to create the mirror, or simply clicking on the horizontal or vertical tools. You can choose to create a copy of the object as well.


Mirror Dialog

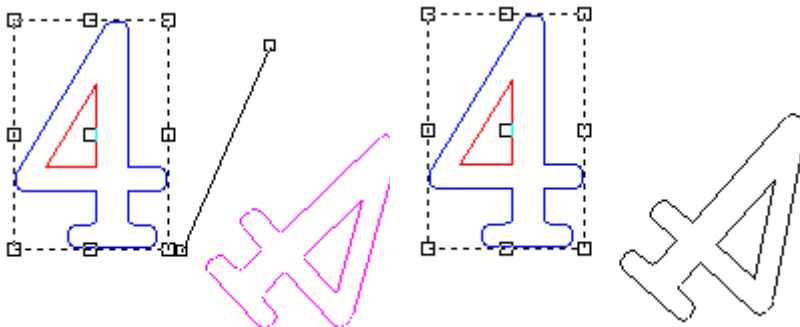
Click the **Mirror Icon**.  This will open the Precision Toolbar.



The Precision Toolbar for the Mirror tool includes a vertical stack of three icons on the left: a mirror icon (selected), a horizontal axis icon, and a vertical axis icon. To the right are input fields for Point 1 and Point 2 coordinates (X, Y, Z). Point 1 X is 3.6993, Y is -1.3180, and Z is 0.0000. Point 2 X, Y, and Z are all 0.0000. Further right is a 3x3 grid of icons for selecting different axes and a 'Copy' checkbox which is currently unchecked. On the far right are 'Apply' and 'Close' buttons.

Mirror Interactively

1. Click the **Mirror Line Icon**. 
2. Select the object or objects that you wish to mirror.
3. Simply click to place the start point of the axis and click again to place the end point. The axis can also be defined by entering the parameters for the start point and the end point of the axis line. A preview of the mirrored object will show on the screen as you are working to place your object.



Without checking the copy box the reversed 4 will be the end result.

If you check the copy box, you will still have the original object as well as a reversed copy.


Mirror Horizontal

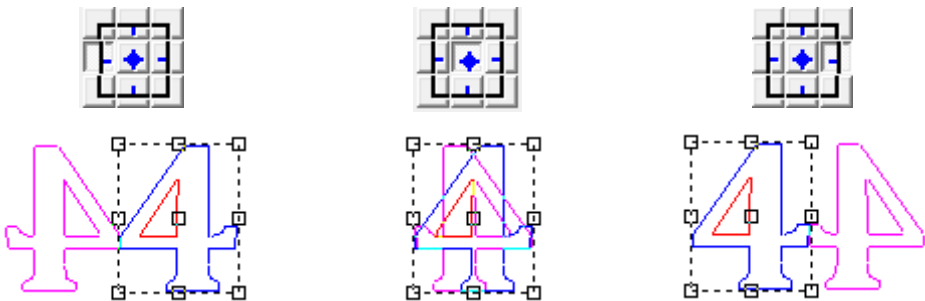
Menu: Transform / Mirror

Toolbar: Object Edit / Mirror

- 1. Click the **Mirror Icon**. This will open the Precision Toolbar



- 2. Select the object or objects that you wish to mirror.
- 3. Click the **Mirror Horizontal Icon**.  A preview of the mirrored objects will show in pink.
- 4. Click the reference grid button that corresponds to side or center point that will be used to locate the mirrored object.



- 5. Click **Apply** to accept.



Without checking the copy box the reversed 4 will be the end result.

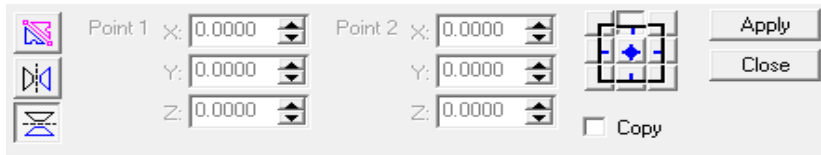
If you check the copy box, you will still have the original object as well as a reversed copy.


Mirror Vertical

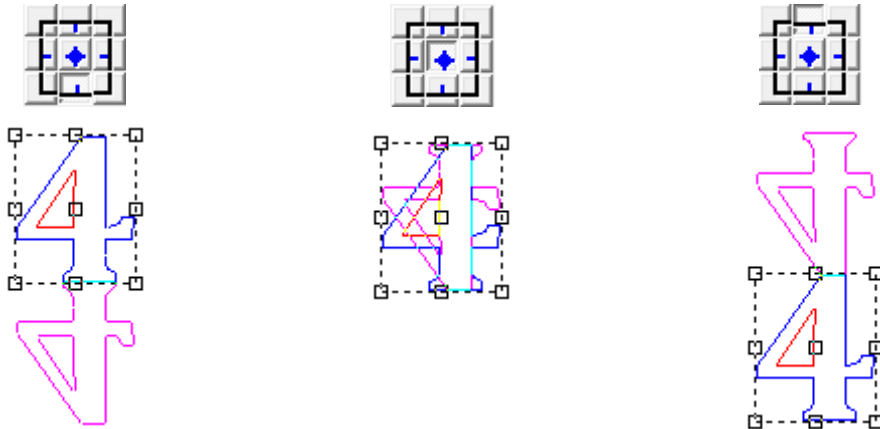
Menu: Transform / Mirror

Toolbar: Object Edit / Mirror

1. Click the **Mirror Icon**. This will open the Precision Toolbar.



2. Select the object or objects that you wish to mirror.
3. Click on the **Mirror Vertical Icon**.  A preview of the mirrored objects will show in pink.
4. Click the reference grid button that corresponds to side or center point that will be used to locate the mirrored object.



5. Click **Apply** to accept.



Without checking the copy box the reversed 4 will be the end result.

If you check the copy box, you will still have the original object as well as a reversed copy.

Aligning Contours

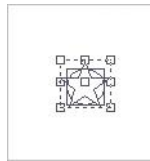
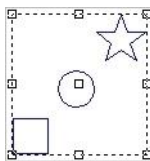
Menu: Transform / Align

Toolbar: Object Edit / Align

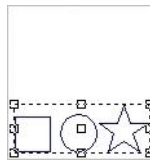
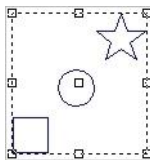
The Align tool allows you to align one or more contours or groups relative to each other.

1. Select the contours to be aligned.
2. Click and hold the **Align Icon**, then one of the following:

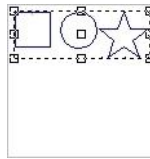
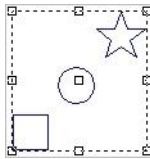
Alignment [to horizontal & vertical center]



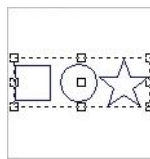
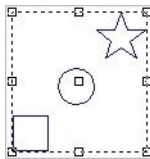
Align to bottom



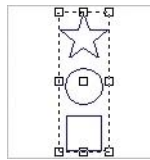
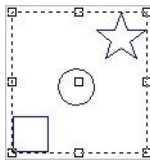
Align to top



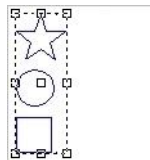
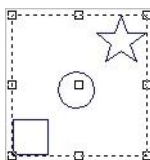
Align to center vertically



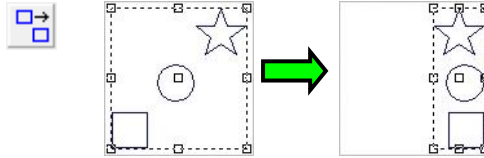
Align to center horizontally



Align left



Align right



Welding Contours

Menu: Transform / Weld

Toolbar: Object Edit / Weld

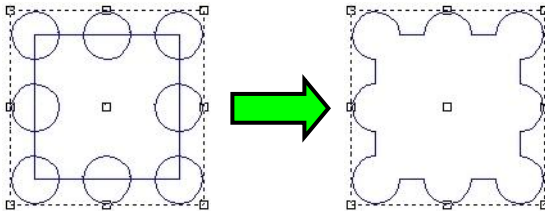
All of the Weld tools work by combining multiple contours into one. Each tool yields different effects, however, due to the way in which the volumes of the contours are combined.

Using the Weld Joined tool

Menu: Transform / Weld / Weld Joined

Toolbar: Object Edit / Weld / Weld Joined

The Weld Joined tool combines multiple contours into one contour. Overlapping areas are merged together to form a contour that is the union of the volumes of all its parts.



To weld contours together using the Weld Joined tool:

1. Select the contours to be welded together.

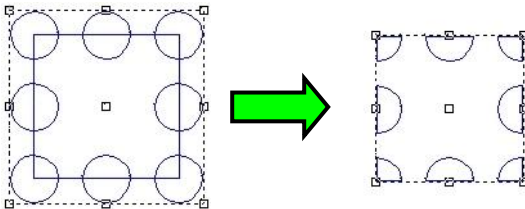
2. Click **Weld joined**. 

Weld Common


Menu: Transform / Weld / Weld Common

Toolbar: Object Edit / Weld / Weld Common

The Weld Common tool combines multiple overlapping contours into one contour that is the intersection of the volumes of the contours.



To weld contours together using the Weld Common tool:

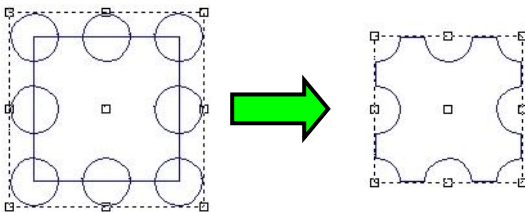
1. Select the contours to be combined.
2. Click and hold **Weld** then click **Weld Common**. 

Weld Subtract


Menu: Transform / Weld / Weld Subtract

Toolbar: Object Edit / Weld / Weld Subtract

The Weld Subtract tool subtracts the volume of a number of contours from the volume of the first contour selected. The result is the first contour, minus any areas which overlapped with the other contours.



To weld contours together using the Weld Subtract tool:

1. Select the contour you want to subtract from.
2. Select the other contours to subtract from the first.
3. Click and hold **Weld** then **Weld Subtract**. 

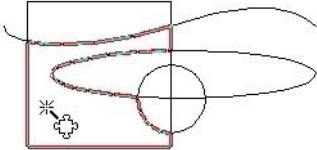
Weld Jigsaw

Menu: Transform / Weld / Weld Jigsaw

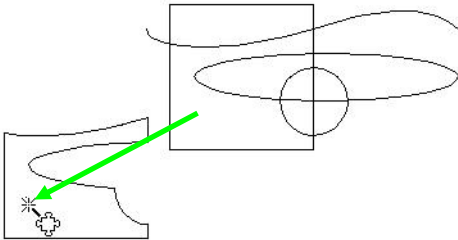
Toolbar: Object Edit / Weld / Weld Jigsaw


The Weld Jigsaw tool allows you to create shapes based on the apparent intersections of contours in your design.


After activating the Jigsaw tool, you can move the cursor over the contours in your design and the software will outline closed shapes that it might create. Contours that are on different levels will act as though they are on the same level.



When you find the closed shape you want, just click in the shape and that closed contour will be created in place in the design. You can then drag to the shape away from the existing contours.



 Note in the example above that the Jigsaw tool works with any type of graphic in the design, including both closed and open contours. This can be very useful when constructing closed shapes for toolpathing. The traditional CAD method of construction would require constructing lines and curves and then trimming them at intersection points, followed by joining them to create a closed shape. That method still works, but the Jigsaw tool can often get to the same result in fewer steps.

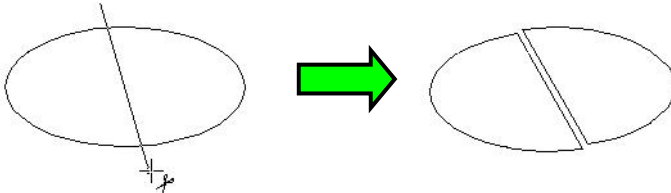
1. Click and hold **Weld** then **Weld Jigsaw**. 
2. Move the cursor over your design to see the contours that can be created by the tool.
3. To create a contour from one of the highlighted outlines, click on the design and drag the new contour away.
4. Right-click to release the Weld Jigsaw tool.


Using the Cut by Line Tool

Menu: Transform / Weld / Cut by Line

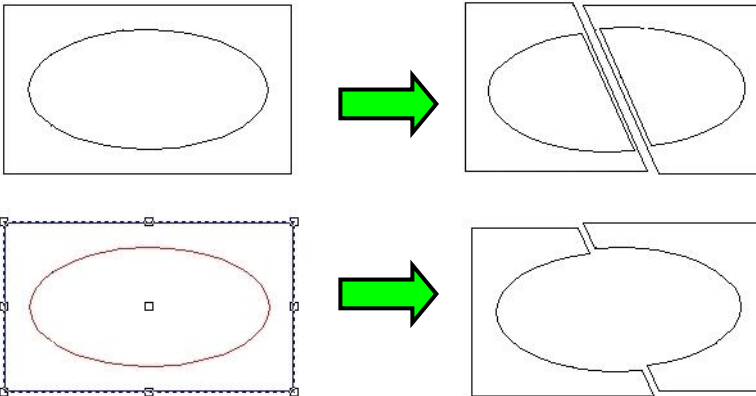
Toolbar: Object Edit / Weld / Cut by Line

The Cut by Line tool allows you to cut closed contours into multiple closed contours.



1. Click **Weld** then **Cut by line**. 
2. Click and drag to create your cutting line.
3. Right-click to release the tool.

The Cut by Line tool operates differently when used on multiple contours that are selected. This is because when you select a group of contours they are automatically grouped into containers and holes. When you use the Cut by Line tool, this grouping is respected. The following images show two contours that are cut, first without being selected, and second while selected. The first example creates four separate contours, while the second example creates just two new contours.



Nesting Contours

Menu: Transform / Nest

Toolbar: Object Edit / Nest

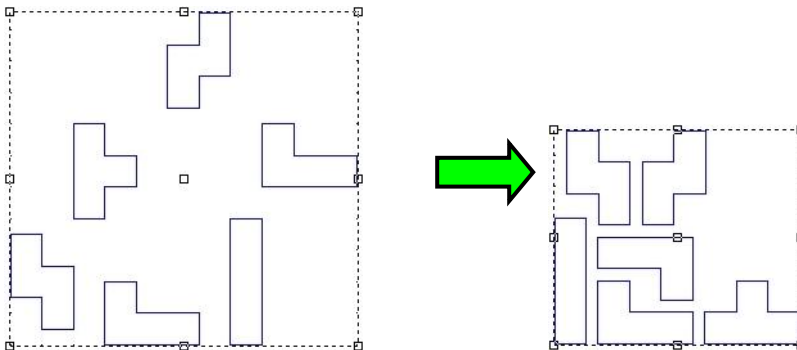
The Nesting tool allows for three different ways of nesting objects and has the ability to nest into non-rectangular shapes and to save shapes that represent partial sheets that can be used for future nests. EnRoute also provides the option to have a nesting report generated that provides information about the efficiency of the nest.


Nest Objects Tool

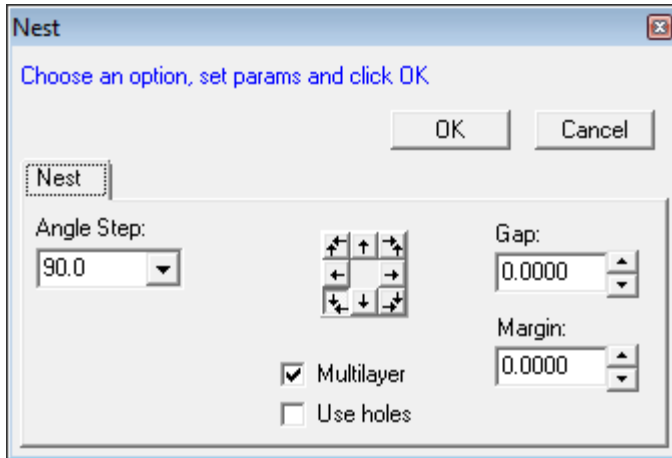
Menu: Transform / Nest / Nest Objects

Toolbar: Object Edit / Nest / Nest Objects

The **Nest Objects** tool automatically nests the selected contours together on the plate so that they use the minimum material while maintaining a defined offset from each other, and from the edge of the plate.



1. Select the contours to be nested.
2. Click **Nest**. 
3. Set the desired options:



Angle Step

This option limits the ways the software can rotate the contours in your selection while nesting them.

- ☞ If you are working with wood or brushed metal, it may be necessary to select no rotation, or limit the rotation to 0 or 180 degrees, in order to keep the grain of the material aligned properly.

Reference Grid

Click on the button that represents the position of the plate that you want the nested items to be located.

Gap

The minimum distance between each of the contours being nested. The gap should be large enough that the toolpaths created from the contours won't overlap each other.

Margin

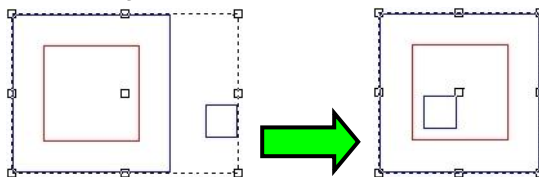
The minimum distance between the edge of the plate and the contours being nested.

Multilayer

If this box is checked, additional layers will be created to allow for the nesting of all selected objects. All of the nested layers will be on and show displayed at the same time immediately after the nest. Click on the arrows in the layers toolbar to display each layer individually and to choose one layer to be active.

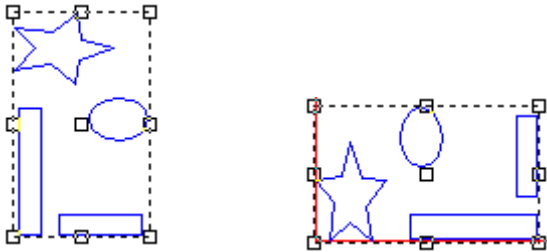
Use holes

If this option is selected, the software will place smaller contours within the holes in larger contours.



4. Click **OK**.


If you have grouped contours together, they will be nested as a group.



Dynamic Nest Precision Toolbar

Menu: Transform / Nest / Dynamic Nest

Toolbar: Object Edit / Nest/ Dynamic Nest

1. Click and hold **Nest** and then **Dynamic Nest**. 
2. Press the **F2** button to open the Precision Toolbar. Once you set these parameters they will not change until you open the Precision Toolbar to change them.

Rotation Steps	<input type="text" value="10"/>	Part Offset	<input type="text" value="0.1000"/>	<input type="button" value="Apply"/>
Nest Resolution	<input type="text" value="0.0100"/>	Plate Margin	<input type="text" value="0.1000"/>	<input type="button" value="Close"/>

Rotation Steps – The rotation steps refer to the highest number of positions you allow EnRoute to try before placing the object.

Nest Resolution – The higher this number is the longer it will take to nest the object and the object will show more clearly.

Part Offset – The minimum distance between each of the contours being nested.


Plate Margin - The minimum distance between the edge of the plate and the contours being nested.

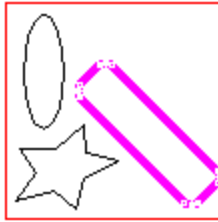
Nesting Tool

Menu: Transform / Nest / Dynamic Nest

Toolbar: Object Edit / Nest/ Dynamic Nest

The **Dynamic Nesting** tool allows the user to drag objects onto the plate and have them automatically rotate to fit within the constraints of other objects and the edge of the plate. Several parameters allow you to adjust offsets and rotation steps.

1. Click and hold **Nest** and then **Dynamic Nest**. 
2. Left click to select the object. It will turn pink.



3. Left click to move the object into position. While you are dragging the object it will turn in several directions as you locate the area best to nest this object.
4. Left click to place the object.
5. Right click to release the object. You can now select another object to nest in the same manner or you can right click again to exit the tool.

Multiple Nest Precision Toolbar

Menu: Transform / Nest / Multiple Nest

Toolbar: Object Edit / Nest/ Multiple Nest

Number	<input type="text" value="3"/>	Gap:	<input type="text" value="0.1000"/>	Priority	<input type="text" value="1"/>	<input checked="" type="checkbox"/> Multilayer	
Angle	<input type="text" value="360.0000"/>	Margin:	<input type="text" value="0.1000"/>			<input checked="" type="checkbox"/> Use holes	

Number – The number of copies that you wish to create and then nest.

Angle – This is the angle in degrees that the objects are allowed to rotate when nesting.

Gap - The minimum distance between each of the contours being nested.

Margin - The minimum distance between the edge of the plate and the contours being nested.

Priority – Enter the number of the priority for each object to be nested to determine the order it is to be nested.


Multilayer – If this box is checked, additional layers will be created to allow for the nesting of all selected objects. All of the nested layers will be turned on. It will show all of the layers displayed immediately after nesting. Click on the arrows in the layers toolbar to display each layer individually and to choose one layer to be active.

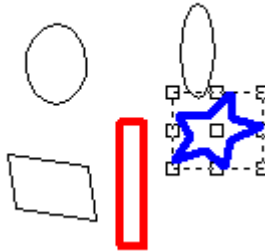
Use holes – If this option is selected, the software will place smaller contours within the holes in larger contours.

Reference Grid - Click on the button that represents the position of the plate that you want the nested items to be located.

Multiple Nest Tool

The **Nest Multiple Objects** tool allows the user to specify the number of copies you want to nest of selected objects. You can also tell the tool to fill a sheet with copies of a selected object.

1. Click and hold **Nest** and then **Multiple Nest**.  The **Precision Toolbar** will open.
2. Select the first object to make copies of and nest. It will turn magenta. Complete the parameters for the nest.
3. Select the next object to nest. The first object will now turn blue indicating that you have already nested this object. The object that you are working with will be the magenta color.



4. Continue to select objects and enter the parameters for each of these objects.
5. When you have finished entering the data for each of the objects that you wish to copy and nest, **Click Ok**. The copies will be placed in the plate.
6. If you have chosen the Multilayer option, there may be more than one layer of nested objects.

8. Working with Toolpaths

The goal of the design process is the creation of toolpaths and the machining of finished pieces. This chapter provides a detailed explanation of each of the different ways to create toolpaths.

Overview

Contours and Toolpaths

Toolpaths derive their shapes from the contours in your design. The contours may form the boundaries of the toolpath, as with a fill, or the toolpath may trace inside, outside or along the path of the contour.

While contours may contain Bezier curves, toolpaths are composed exclusively of arcs and straight line segments, in order to be compatible with the movements of tools in the routers, engravers and other and CNC devices which are used with the software. Any Bezier curves which may be present in a contour are replaced by combinations of straight lines and arcs in the toolpath.

Toolpath Groups

When a toolpath is generated from a contour, both the toolpath and the original contour are grouped together into a *toolpath group*. If additional toolpaths are created based on the same contour, they will be added to the same toolpath group.

Each contour can belong to one and only one toolpath group. If you want to generate a toolpath based on a contour by itself, and a different toolpath based on a group of contours which includes the first contour, you will need to make two copies of the contour: one to use by itself, and one to use as part of the group.



When this is necessary, it is helpful to keep the individual contour and the group of contours on separate layers in order to avoid confusion.

Strategy

When creating a toolpath, you must choose a *strategy* which determines the type of toolpath the software will generate (fill, engraving, slot, etc.). Each strategy has its own set of parameters which define virtually every aspect of the toolpaths that are created, such as tool type, depth, overlap, direction and speeds.

Because the strategy completely determines the appearance and shape of the final toolpath, the software always refers to the toolpath by the strategy that was chosen, not as a toolpath.

Plan

The *plan* is the set of all of the different strategies contained in any one toolpath group.

Cuts

Each toolpath strategy is made up of one or more cuts. For instance, a fill might have a Rough cut, a Fine cut, and a Clean cut. Each cut uses one specific tool, as well as the parameters associated with the type of cut it will be making.

Plan, Strategy and Cut Templates

At each step in the toolpath creation process, it is possible to define and save a library of templates that can be used to automate the process.

- ⌚ A cut template saves the parameters associated with a single cut.
- ⌚ A strategy template saves the parameters associated with a strategy, including the parameters of each of the cuts that make up the strategy.
- ⌚ A plan template saves all of the parameters associated with all of the strategies for a given toolpath group, as well as the parameters for each of the cuts in each of the strategies.

The information saved in the templates can then be easily selected and reapplied to a different contour, recreating all the work that was done on the original toolpath group in one powerful step.

For example, a common activity is to create male and female routing offsets to be used to cut out different materials so that one may be inlaid into another. Both sets of toolpaths can be created based on one set of contours, and then selectively used to cut out the material. After determining the best combination of parameters for both types of offsets, they can be saved as a plan template so that these parameters may be recalled and reapplied at any time.

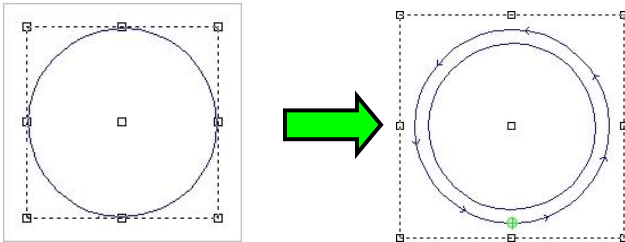
Creating Routing Offset Toolpaths

Menu: Toolpath / Routing Offset

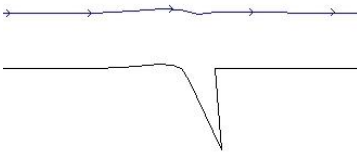
Toolbar: Toolpath / Routing Offset

📄 The Routing Offset strategy can only be used with closed contours.





The Routing Offset strategy creates a toolpath that runs either inside or outside the selected closed contour. The toolpath is offset from the contour by the radius of the cutting tool, so that it will trim right up to the edge of the contour.



If the cutting tool cannot fit into a corner or tight curve of the contour, the toolpath will follow the closest smooth curve possible.

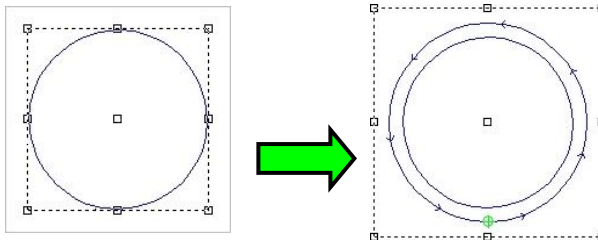


To create a toolpath using the Routing Offset strategy:

1. Select the contour you want the toolpath to follow.
2. Click **Routing Offset**. 
3. In the Routing Offset dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 -  The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
 -  The first tool in the list is always defined as the Rough tool, and it is typically used as the main cutting tool. Any additional tools serve as Clean or Fine tools, primarily used as a final pass to clean up cutting marks from the Rough tool, or to fit into areas that the Rough tool couldn't.
4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the cut.
 -  It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.
5. Add additional cuts using other tools as necessary.
6. Set the Router Offset parameters:

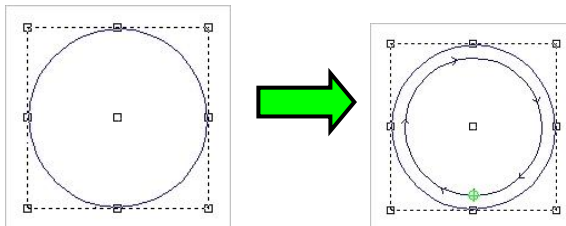
External (male)

If checked, the toolpaths will follow the outside of the contour.



Internal (female)

If checked, the toolpaths will follow the inside of the contour.




Weld offsets If checked, overlapping offsets will be welded together to form a single toolpath group. If cleared, overlapping offsets will be left the way they are.

Sharp corners

When toolpaths are offset from square corners, they are naturally rounded. This does not affect the 'squareness' of the corner and it generally allows for smoother movement of the machine. If this option is checked, the toolpath

offsets form square corners. It is generally not recommended to choose this option.

Inlay If checked, the toolpath defines either the socket for an inlaid piece of a different material (if the toolpath is set to **Internal**), or the cut that will separate the inlay itself from the plate (if the toolpath is set to **External**).

 Because a round bit is being used to cut out both pieces, both the male and female toolpaths need to take into account the dimension of the tool. This changes the shape of the toolpath, particularly in corners.


If this option is checked, the **Inlay Gap** parameter displays. This parameter indicates the size of the gap that will exist between the inlay and its socket.

With Bridges Bridges are small tabs of material that are left connecting parts which would otherwise be completely severed from the rest of the design. After the job has finished cutting, the operator can then manually cut the bridges to remove the part from the rest of the plate.

If this box is checked, bridges will be created.

If the **With Bridges** option is checked, other parameters are activated to allow the user to enter the number, size and spacing of the bridges.

7. Click **OK**.

 Once you create the toolpath, the contour will become part of the toolpath group.

Bridges

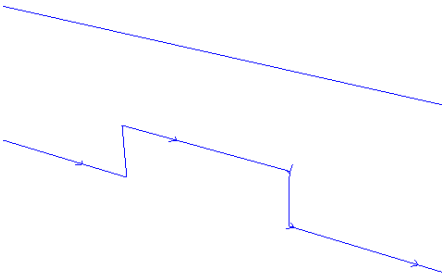
Bridges are lifts in a routing offset toolpath that create a tab that maintains the connection between a part that is being cut out and the material from which it is being cut. This tab helps increase the stability of the part so that it doesn't move as it is being cut. The tabs can then be trimmed away from the part. EnRoute provides several different types of bridges so that you can decide which bridge shape works best for the material type and thickness you are cutting.

After choosing the **With Bridges** option in routing offset parameters dialog, you have several parameters to define for the bridges, including length, height, type and number. The following table describes the bridge parameters.

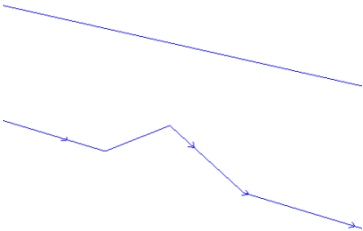
Type	This provides the option to select the shape of the bridge. You will probably find a shape that works best with the material you cut. The smooth shapes have the advantage of allowing the machine to move smoothly through the bridge.
Length	This is the length of the bridge. Shorter bridges will work in strong material. If you are cutting soft material you may need to increase the length of the bridge.
Height	This also lets you adjust the bridge size to fit the material.
By Number	This option lets you choose the number of bridges in the offset.
By Distance	This option lets you choose the spacing of the bridges.
Manual	Rather than specifying the number or spacing of the bridges, you may want to define the specific location of the bridges. Remember though, that even if you specify the bridges by number or distance, you can easily edit the location of the bridges after they are created.

This shows the different types of bridges that are available in EnRoute.

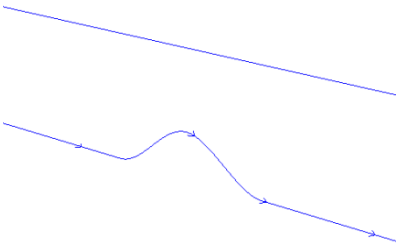
Lift



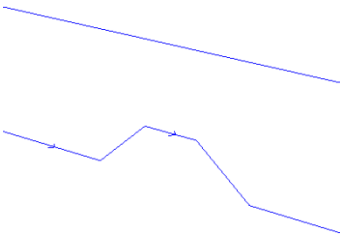
Ramp



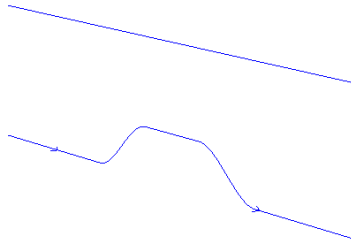
Smooth



Ramp Mesa



Smooth Mesa



Deleting Cuts

To delete an unwanted cut from the list of cuts for the strategy, select the cut and click **Delete tool**.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

Editing Cut Parameters

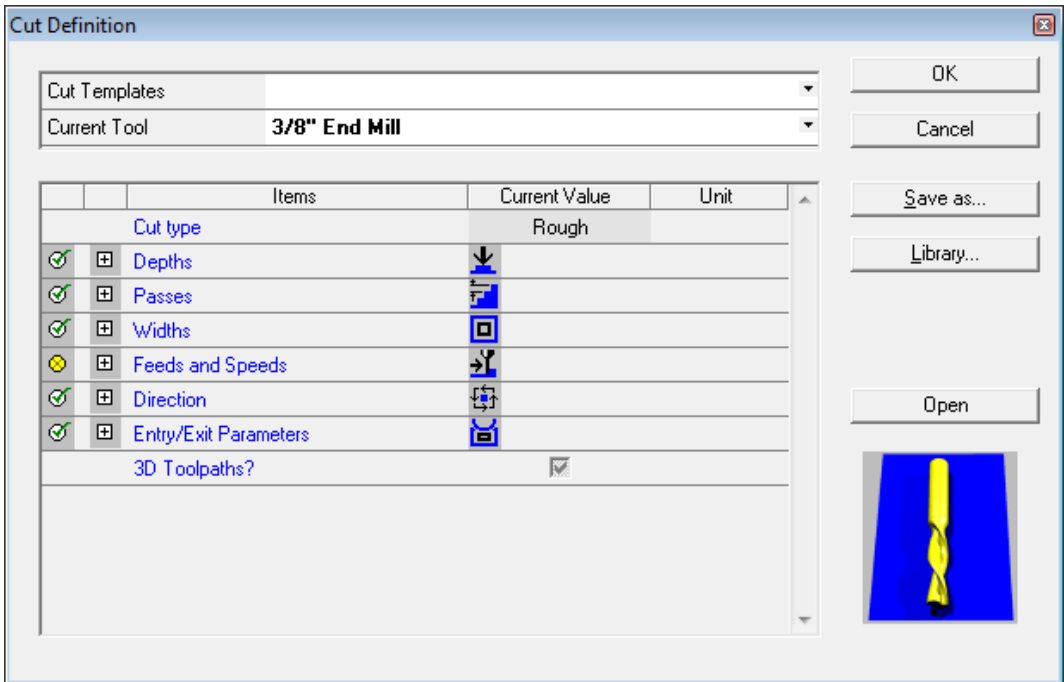
The Cut Parameters dialog allows you to fine-tune the cuts that make up each strategy by changing the cutting parameters from the default values.



For example, if you are cutting a hard material such as aluminum, it likely will be necessary to define a greater number of passes that will use a much smaller portion of the available cutting depth of the tool on each pass.

To edit the parameters for a cut:

1. Select the cut in the cut list and click on the ellipses in the **Edit** column.
2. Edit the cut parameters.
3. Click **OK**.

The following figure shows the Cut Parameters dialog for the roughing tool in the Routing Offset strategy.



The cut parameters are organized into an explorer list. Click on  or  to expand or collapse an entry in the list.

To the left of the opened/closed symbol is an indicator about the readiness of the parameter values in that category. A green check mark indicates that the values are all 'legal'. A yellow symbol with a green X indicates that the values may not be entirely complete, but that this won't stop the strategy from being created successfully. A red symbol with an X indicates that one or more of the values in that category make it impossible to successfully create the strategy. When the red symbol is displayed, the **OK** button in the dialog box is also disabled.

Cut Parameter Definitions

Current Tool To change the tool that is defined for this cut, select the new tool from the **Current Tool** list.

Cut Type This shows the cut type of the current cut being edited. In some strategies, the cut type may be changed at this location.
In the Routing Offset strategy, the cut type is either **Rough**, for the first cut in the list, or **Clean** for any subsequent cuts that are defined.

Depths

Surface Depth This is the distance from the top of the plate to the top of the cut. A large portion of the time this will be left at the default value of 0.00. There are times, however, when it may be appropriate for the Surface to be set down from the top of the plate, possibly if this cut is to be performed after another cut that has removed part of the material depth.

Final Depth This is the depth of the cut, measured from the Surface depth. If the Surface depth is set to 0.00 then the Final Depth is measured from the top of the plate. If the Surface depth is greater than 0.00 then the Final Depth will be measured starting from the Surface Depth. For example, a Surface Depth of 0.50 and a Final Depth of 0.50 would produce an actual cut depth of 1.00.

Passes

Number This shows the number of passes that will be created. EnRoute calculates the minimum number of passes required based on the Final Depth and the Max per Pass that is allowed for the selected tool. It is not possible to define a number of passes less than the minimum, but it is quite common to define the number of passes greater than the minimum in order to reduce the amount of material that is removed with each pass.

Max Per Pass This value is initially calculated based on the dimensions of the selected tool. However, it is possible to reduce this value in order to control the depth of each pass.

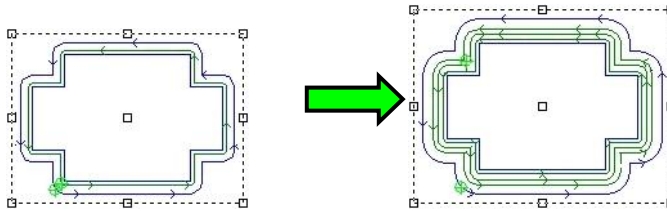
Actual Per Pass This value is calculated based on the number of passes and the Final Cut depth. If a Final Pass is defined, the amount of the final pass is removed from the calculation of the Actual Per Pass value.

Final Pass If the number of passes is greater than 1, it is possible to designate a Final Pass that will have a cut depth that is different from the other passes. A common use for this is to cut mostly through the material with the passes preceding the Final Pass, and reserve a small thickness that will be cut with the final pass to separate the pieces being cut from the substrate.

Final Pass Depth This is the depth of the Final Pass. It will obviously be a depth that is less than the Final Depth of the entire cut.

Widths


Width of Cut In addition to the cut depth, it is also possible to define a Width of Cut. This is most commonly defined with a Clean Cut, but it may be defined for either a Clean or Rough-cut. If a Clean cut is defined with a 0.00 width, it will essentially be located at the same offset from the contour as the Rough cut. It would 'clean' by making another pass next to the rough pass to provide a smoother edge, or to possibly fit into areas that the rough tool could not. If a width of cut is defined for the Clean cut, then the Rough cut is offset away from the finished surface by the amount of the width. The Rough cut would leave that width of material that would then be removed by the Clean cut. The following toolpath images illustrate this concept.



Number of Steps	This is similar to the concept of Passes with cut depth, except it is the diameter of the tool that controls the Number of Steps that is required for a given Width of Cut. The Number of Steps is calculated automatically based on the Max Step value, and the Width of Cut.
Max Step	This is the maximum width of material that can be removed in one step. It is based on the diameter of the tool being used. The software automatically calculates this value to be 90 percent of the tool diameter. The value can be adjusted so that it is smaller than this, but this is the largest it is allowed to be.
Actual Step	The Actual Step is calculated based on the Number of Steps and the Width of Cut values.
Shoulder	<p>This is an option that provides a way to create a step around the perimeter of the design. The purpose of this option is to allow users to create a 'lip' around the edge of the design that is most commonly used with the Inlay option with Routing Offsets and Fills.</p> <p>Any Clean Pass may be designated as a Shoulder, but it must have a Width of Cut that is larger than 0.00 in order to be designated as a Shoulder. When the Shoulder option is selected, all Clean Pass cuts below it in the list will be positioned out from the Shoulder pass.</p>

Feeds and Speeds

This section of the Cut Parameters allows you to specify the feed rates that will be used with this cut. If the tool you are using has feed rates saved for the material you are using, these values will initially reflect the saved values. The units used in the feed rates are initially set to match the values for speed in Preferences Unit section. However, you can change the units to be different if there is a need.

-  If your plate has a material selected, the feed rates for that material become the default feed rates for new cuts. If the tool in your cut has feed rates assigned for your material, these feed rates override the material's feed rates. If no material has been assigned, then the initial feed rates are 0.00.

Feed Rate	This is the speed that the tool will move in the material when it is cutting.
Final Pass Feed	If the cut is using the Final Pass option, this value represents the speed for this pass.
Plunge Rate	This is the speed that the tool will move down into the material.
Dwell	After plunging into the material, a dwell value will cause the cutting tool to pause for this length of time before feeding into the material.

Spindle Speed This is the speed that the spindle will turn while it is cutting.

Direction

Conventional This direction will move the tool 'with' the direction that the tool is turning. For a standard exterior cut on a contour, this would be in a counter-clockwise direction.

Climb This is the opposite of conventional. The tool will move 'against' the direction that the tool is turning. For an exterior cut, this is in a clockwise direction.

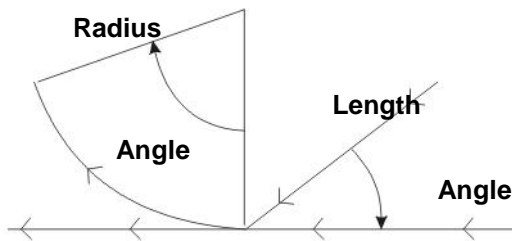
Entry/Exit Parameters

When a tool begins and ends its cut, it is sometimes desirable to have it move into and out of the cut away from the actual finished edge. The entry and exit provide the means to do this. Each of these may be specified separately; you can have an entry that is an arc, and an exit that is a line.

It is also possible to specify either an arc or line as being '3D'. With an entry, this means that the tool will start at the surface depth and progress down to the cut depth over the length of the entry. The exit does the same thing only in reverse. It starts at the end of the cut and progresses up to the surface over its length. With an arc, the arc becomes a helical arc if it is set to 3D.

📄 If you specify an entry and/or exit as a line, and then enter an angle of 0.0 degrees, the entry/exit will follow the toolpath, even if the toolpath turns over the length of the entry/exit. This can be useful if toolpaths are close together and you want to make sure they don't interfere.

The diagram below depicts how the dimensions of the entry and exit are specified.



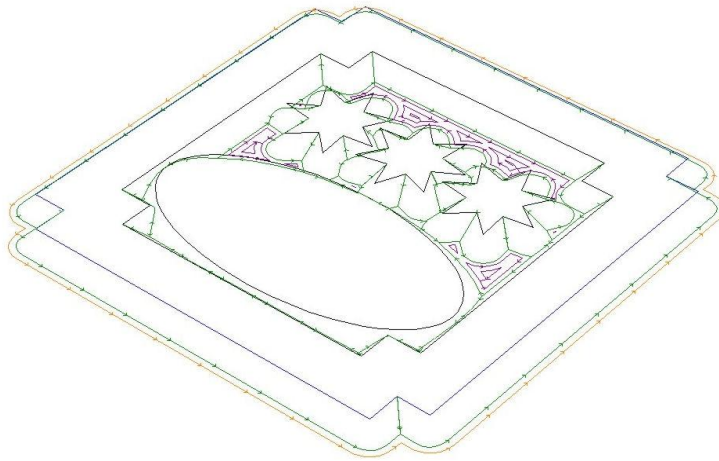
3D Toolpaths

This option designates whether this cut will be a 3D pass or not. A 3D toolpath may be created using a conic, engraving or tapered tool. The difference is that it uses the beveled shape of the tool to create a beveled edge. It also will move up into corners to get as far into the corners as the tip dimension of the tool will allow.

As you can imagine, a 3D Toolpath designation is only appropriate in certain situations, and with certain tools. The software monitors the type of cut and the type of tool that has been specified, and then controls when the 3D Toolpaths option may be selected. Also, this option is not available at all licensing levels of the software. If your version does not have the 3D Toolpaths option enabled, it is possible to enable this function by upgrading the software license.

The 3D option is a very powerful option, allowing a very different type of cut from any of the toolpaths that are strictly 2D. 3D toolpaths may be created for cleaning cuts with Routing Offsets, Island Fills, and Hatch Fills. With the Engrave Strategy, the 3D toolpath will likely serve as the primary option on many jobs.

The following figure provides an illustration of some of the 3D toolpaths that may be created.



Saving Cut Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Cut Settings from a Template

To load the settings previously saved as a template, select the template from the **Cut** list.

It is possible to load a cut template that was defined for one Strategy type into the Cut Parameters dialog for a different strategy type. Since different strategies use different cut parameters, it is possible that not all of the parameters for the new cut will be defined in the template.

Creating Open Contour Offset Toolpaths

Menu: Toolpath / Offset Open Contours

Toolbar: Toolpath / Routing Offset/ Router Offset Open Contour


The Open Contour Offset is closely related to the Routing Offset strategy described in the previous sections. The key difference is that the Routing Offset strategy is designed to be used with closed contours, and Open Contour Offset is designed for open contours.


To create a toolpath using the Open Contour Offset strategy:

1. Select the contour you want the toolpath to follow.


2. Click and hold **Routing Offset**  then **Offset Open Contour**. 

3. In the Open Contour Offset dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.

 The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.

 The first tool in the list is always defined as the Rough tool, and it is typically used as the main cutting tool. Any additional tools serve as Clean tools, primarily used as a final pass to clean up cutting marks from the Rough tool, or to fit into areas that the Rough tool couldn't.

4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the cut.

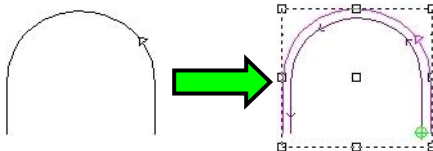
 It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.

5. Add additional cuts using other tools as necessary.

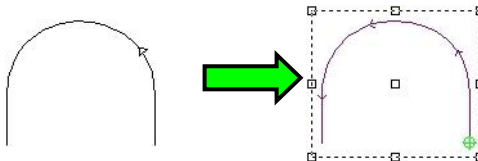
6. Set the Open Contour Offset parameters:

Side Rather than defining whether the offset is to be internal or external as with the Routing Offset, it is necessary to define which side of the contour to place the toolpath. **Left** and **Right** are determined as if you are facing in the direction of the contour. If it is located in the middle, the toolpath is placed on top of the open contour.

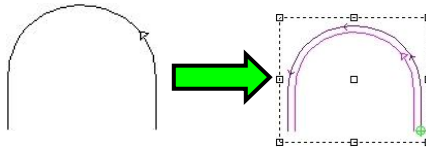
Left



Middle



Right

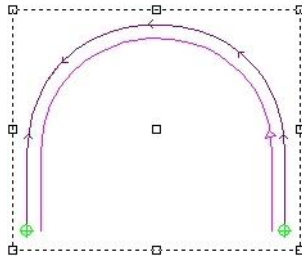


With Relief

One of the common uses of an Open Contour Offset is to cut out sections around the perimeter of a design. For example, you might use a standard routing offset to cut out a simple rectangle, and then use an Open Contour Offset to cut out additional shapes around the outside of the rectangle. In certain materials, and especially wood, it is necessary for the tool to move into the material rather than moving from inside the material out in order to minimize the risk of the material 'blowing out' as the tool leaves the material.

The **With Relief** option allows you to define a segment of the toolpath at its end to move in rather than out so that you can ensure that the toolpath won't move the tool out of the material as it is cutting.

The graphic below illustrates an open contour that uses Relief at the end. The left end of the toolpath is the 'end' with Relief. Notice the direction arrow pointing the other way: this is part of the Relief section.



If the **With Relief** option is selected, the following additional parameters are displayed.

Relief Length	This is the length of the section at the end of the toolpath that will move back in from the end.
Relief Overlap	It is possible to overlap the two sections to ensure a clean cut.
Cut End First	If this is selected, the Relief section of the toolpaths will be cut before the rest of the Open Contour Offset toolpaths.

7. Click **OK**.

 Once you create the toolpath, the contour will become part of the toolpath group.

Deleting Cuts

To delete an unwanted cut from the list of cuts for the strategy, select the cut and click **Delete tool**.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

Editing Cut Parameters

The process for editing cut parameters for the Open Contour Offset strategy is nearly identical to that for the Routing Offset strategy (see “Editing Cut Parameters”).

The two exceptions to this are:

- It is not possible to define the toolpaths as 3D toolpaths so the **3D Toolpath** option is not available.
- The direction of the toolpath is defined by the direction of the contour, so the **Direction** cut parameter is not available.
- Note that you are able to define widths for each type of cut. This allows you to use open contours to cut out areas that are wider than the tool you are using to perform the cut.

Creating Hatch Fill Toolpaths

Menu: Toolpath / Hatch Fill

Toolbar: Toolpath / Hatch Fill

- The Hatch Fill strategy can only be used with closed contours.

The Hatch Fill Strategy is used to create toolpaths to mill a surface down to a defined depth using toolpaths that move back and forth across the area to be milled. This option is used most often to mill out larger areas of material that typically don't have a lot of thin sections, but it can be used on any artwork.

Types of Cuts

There are three types of cuts that can be defined in the Hatch Fill Strategy, the Fill Cut, Fine Cut, and Clean Cut. Each of these is explained below.

Fill Cut

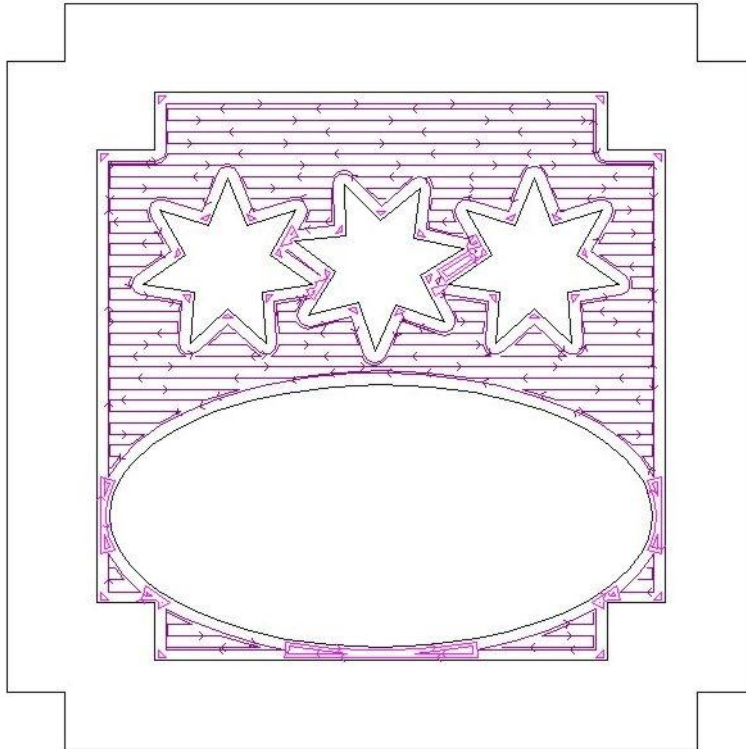
This is the tool that will perform the Hatch Fill. You will almost exclusively use an end mill tool to perform the fill. In addition to the standard cut parameters that have been described previously, with the Fill Cut it is necessary to define two additional key parameters – **Overlap** and **Angle**. See “Editing Cut Parameters” for details.

Fine Cut

The Fine Cut is optional. If it is used, a tool that is smaller in diameter than the Fill tool is selected to

fit into sections of the design that the fill tool was not able to fit into. This will be corners and any other thin areas that are too small for the Fill tool. The Fine Cut uses an Island Fill strategy to fill in areas that require more than one width of the tool. For this reason, it is also necessary with the Fine Cut to specify the amount of overlap between adjacent toolpaths.

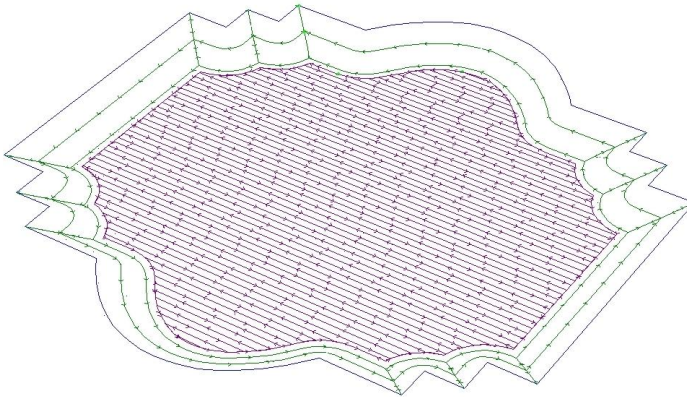
The following screen shot shows a Hatch fill with a Fine Cut used to fit into the areas where the fill tool won't fit.



Clean Cut

The final toolpath option with the Hatch fill is the Clean Cut. This defines a tool that creates toolpaths that are offset from the contours. The Clean Cut is used to improve the edge quality of the finished cut, or to create a 3D toolpath around the perimeter of the design.

The following image shows a Hatch fill that includes a 3D Clean Cut. This combination of using a fill pattern along with a 3D clean cut is a very common technique used in many engraving, woodworking and sign-making applications. Many times there is a need to utilize a 3D engraving toolpath, but the material doesn't permit the 3D toolpath to go to the 'bottom' of the design. In this case, the fill can be used to define the bottom of the design.





To create a toolpath using the Hatch Fill strategy:

1. Select the contour you want the toolpath to follow.


2. Click **Hatch Fill**. 

3. In the Hatch Fill dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.

 The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.

 The first tool in the list is always defined as the Rough tool, and it is typically used as the main cutting tool. Any additional tools serve as Clean tools, primarily used as a final pass to clean up cutting marks from the Rough tool, or to fit into areas that the Rough tool couldn't.

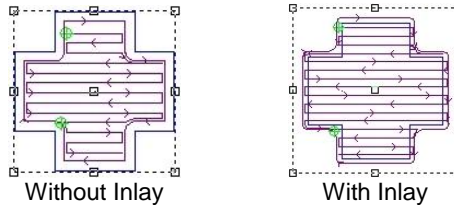
4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the cut.

 It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.

5. Add additional cuts using other tools as necessary.
6. Set the Hatch Fill parameters:

Inlay

This defines whether this strategy is part of an inlay project. If this fill will be used as the female part of an inlay project, it is necessary for the software to modify the toolpaths to compensate for the dimensions of the tool. Also, you will need to define the inlay gap, which is the spacing between the male and female portions of the inlay. A gap is needed to allow the two parts of the inlay to fit together. The screen image below shows two contours that have been hatch filled. The contour on the right has also been defined as an inlay to illustrate how the toolpaths are modified at the corners.



Optimization

There are three different options regarding the way in which the hatch toolpaths are ordered within the contour. In the machining industry there are many different viewpoints regarding how 'pocket milling' toolpaths should be ordered in order to be most efficient. The software utilizes a method that is intended to minimize tool lifts in the fill, but not necessarily to prevent tool lifts.

In many cases it is not practical to attempt to eliminate tool lifts without breaking other rules that the software enforces. The software does not allow any of the hatch toolpaths to pass over any part of the hatch area more than one time. Also, it strictly enforces a back-and-forth pattern over the entire hatch area. We feel that this results in a more aesthetically pleasing finish on the hatch surface, and it is consistent with methods employed by other machining packages.

A second important option for ordering hatch toolpaths is to make them compatible with a specific type of engraving tool that utilizes a 'Nose Cone' that maintains a constant depth of cut for the tool. This requires that the hatch toolpaths progress from one side of the fill to the other, always maintaining an edge of uncut material next to the nose cone.

Finally, the software provides the option of having the cleaning pass that is associated with the hatch toolpath be cut before the hatch toolpaths. This can be a desirable option when cutting certain types of material, such as certain woods.

7. Click **OK**.

 Once you create the toolpath, the contour will become part of the toolpath group.

Deleting Cuts

To delete an unwanted cut from the list of cuts for the strategy, select the cut and click **Delete tool**.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

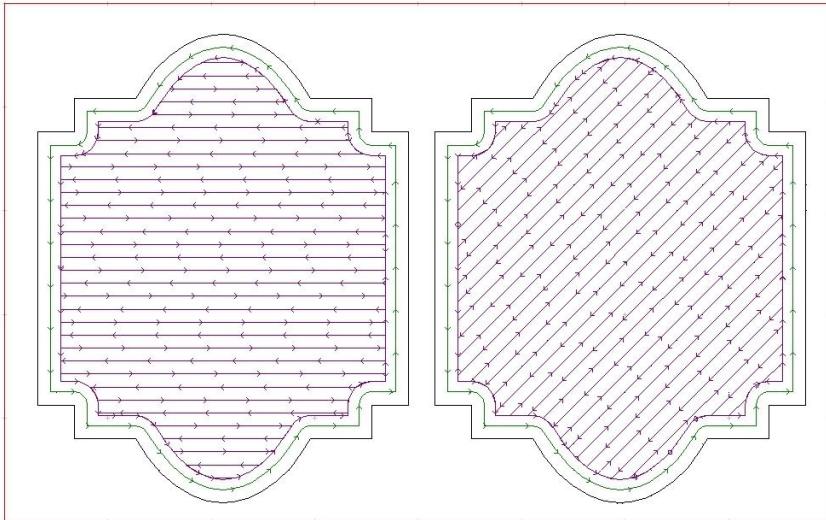
To load the settings previously saved as a template, select the template from the **Strategy** list.

Editing Cut Parameters

The process for editing cut parameters for the Hatch Fill strategy is nearly identical to that for the Routing Offset strategy.

The one key difference lies in the parameters for the Fill cut, where two additional parameters are defined under **Fill**:

- ⌚ **Overlap** – This parameter defines how much adjacent toolpaths will overlap each other in the fill. The default value that is created when the Fill Cut is created is 50 percent. Legal values are 0 to 99 percent. When cutting soft materials you may typically reduce the overlap 10 or 20 percent. However, when cutting denser materials, it may be desirable to increase the overlap percentage to 70 percent or more. You will also notice that the overlap percentage has an effect on the finished surface pattern, so you may at times use the overlap to create a desired effect.
- ⌚ **Hatch Angle** – This angle determines the angle (measured from horizontal) that will be used to create the Hatch Fill. Most of the time you will likely use either 0-degrees for horizontal or 90-degrees for vertical toolpaths. Other angles can be used to create an effect, or to better utilize the shape of the area to be milled.



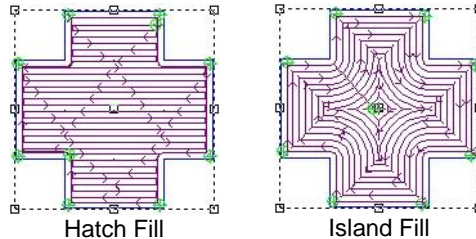
Creating Island Fill Toolpaths

Menu: Toolpath / Island Fill

Toolbar: Toolpath / Island Fill

The Island Fill strategy can only be used with closed contours.

The **Island Fill** Strategy is very similar to the **Hatch Fill** Strategy. The way that the two are defined is virtually identical. The results, however, are quite different because of the pattern of the fill toolpaths that are created.



Island Fill vs. Hatch Fill

There are two main factors to be considered when choosing a fill strategy. First, select the strategy that will most efficiently mill the area to be milled. Typically, if you have a large open area, a Hatch Fill will be most efficient. On the other hand, if you have a design that is made up of long, thin sections, an Island Fill will most likely work best.

The second reason for choosing one method over the other is based on whether the tooling pattern that will be left in the milled area is important. As you work with these strategies, you will see that each of them will create a different pattern, and there may be times when this pattern is important in deciding which strategy is appropriate.

The best thing is to experiment with each and then decide for yourself which method works best in any given situation.


Creating an Island Fill


To create a toolpath using the Island Fill strategy:


1. Select the contour you want the toolpath to follow.

2. Click **Island Fill**. 

3. In the Island Fill dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.

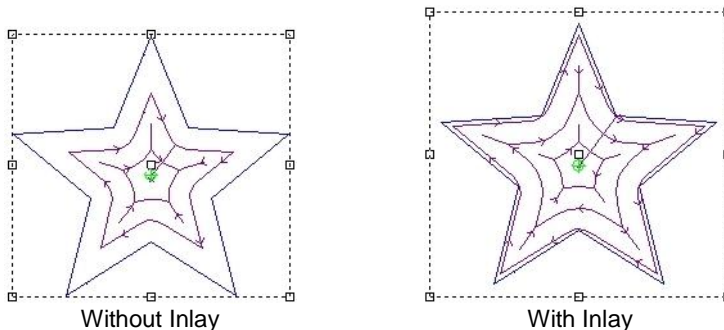
 The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.

 The first tool in the list is always defined as the Rough tool, and it is typically used as the main cutting tool. Any additional tools serve as Clean tools, primarily used as a final pass to clean up cutting marks from the Rough tool, or to fit into areas that the Rough tool couldn't.

4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the cut.
-  It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.
5. Add additional cuts using other tools as necessary.
6. Set the Island Fill parameters:

Inlay

This defines whether this strategy is part of an inlay project. If this fill will be used as the female part of an inlay project, it is necessary for the software to modify the toolpaths to compensate for the dimensions of the tool. Also, you will need to define the inlay gap, which is the spacing between the male and female portions of the inlay. A gap is needed to allow the two parts of the inlay to fit together. The screen image below shows two contours that have been hatch filled. The contour on the right has also been defined as an inlay to illustrate how the toolpaths are modified at the corners.



Optimization

There are three different ways in which the fill toolpaths can be ordered within the contour. In the machining industry there are many different viewpoints regarding how 'pocket milling' toolpaths should be ordered in order to be most efficient. The software utilizes a method that is intended to minimize tool lifts in the fill, but not necessarily to prevent tool lifts.

In many cases it is not practical to attempt to eliminate tool lifts without breaking other rules that the software enforces. The software does not allow any of the hatch toolpaths to pass over any part of the hatch area more than one time. Also, it strictly enforces a uniform pattern over the entire hatch area. We feel that this results in a more aesthetically pleasing finish on the hatch surface, and it is consistent with methods employed by other machining packages.

A second important option for ordering hatch toolpaths is to make them compatible with a specific type of engraving tool that utilizes a 'Nose Cone' that maintains a constant depth of cut for the tool. This requires that the hatch toolpaths progress from one side of the fill to the other, always maintaining an edge of uncut material next to the nose cone.

Finally, the software provides the option of having the cleaning pass that is associated with the hatch toolpath be cut before the hatch toolpaths. This can

be a desirable option when cutting certain types of material, such as certain woods.

7. Click **OK**.

 Once you create the toolpath, the contour will become part of the toolpath group.

Deleting Cuts

To delete an unwanted cut from the list of cuts for the strategy, select the cut and click **Delete tool**.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

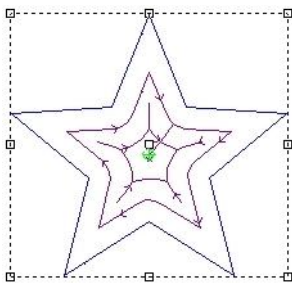
Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

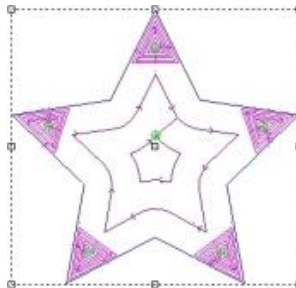
Editing Cut Parameters

The process for editing cut parameters for the Hatch Fill strategy is nearly identical to that for the Routing Offset strategy (see “Editing Cut Parameters”).

One of the options available in the cut parameters for the fill tool in the Island Fill strategy is whether to use Corner Tags. Corner tags are used to ensure that the specified overlap value is enforced throughout the fill. This can be very important when you use an overlap of less than 50 percent, because it ensures that the entire fill area gets milled. The following graphic illustrates a fill toolpath with and without corner tags.



With corner tags



Without corner tags

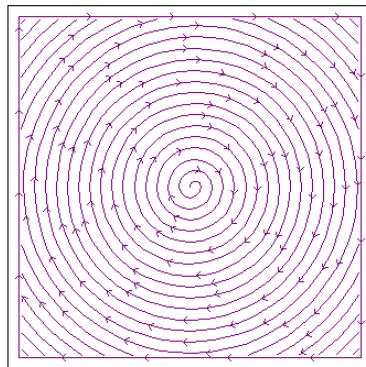
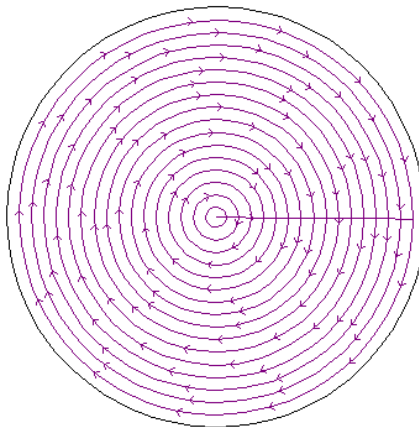
Spiral Fill Toolpaths

Menu: Toolpath / Spiral Fill

Toolbar: Toolpath / Hatch Fill / Spiral Fill

The Spiral Fill strategy is essentially the same as the Island fill, except that it creates a spiral pattern for filling the selected contours. This can be very useful for certain design elements, but it is typically only useful for objects that are round or near round. On round contours it allows the toolpath to be created as a single toolpath. This eliminates the need for the tool to step in after every trip around. The advantage is that this is more efficient, and it can also improve the aesthetics of the tooling marks that are left on the milled surface.

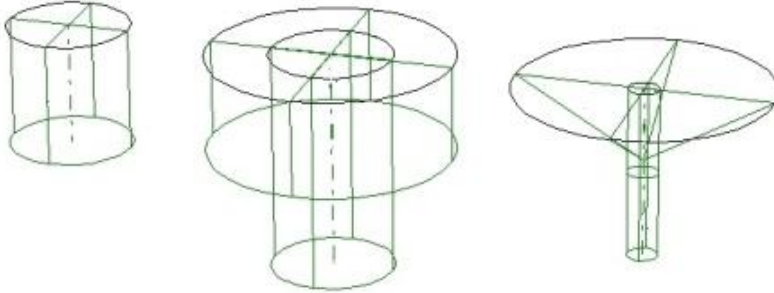
On objects that are not round, the toolpath must stop where it intersects the edge of the part, which requires the tool to lift up and then move to the next portion of the toolpath. This could require many more tool lifts than would be necessary using either a standard Hatch or Island fill.



Drill Tools

Menu: Toolpath / Drill

Toolbar: Toolpath / Drill



Drill points are similar to toolpaths, but they are not the same. Many CNC machines are able to respond differently to drill commands than they do to standard toolpath commands. For this reason, the software creates drills as a different type of object so that when this information is sent to the machine, the driver is capable of determining how much drill information a specific machine requires, and how much should be controlled by the machine. For the most part you don't have to think about drills at this level, but it is important to know that they differ from toolpaths.

There are six different strategies used to create drill points. Each of these strategies is identical in the way that they create the drill points, and in how Drill Cut Parameters are defined. They differ in how they arrange the drill points.




Creating Drill Points

Menu: Toolpath / Drill / Drill Point

Toolbar: Toolpath / Drill / Drill Point

The Drill Point strategy creates a single drill point at the location specified.

To create a drill point:

1. Click **Drill** then **Drill point**. 
2. In the Drill Point dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 -  The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
3. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the drill hole.
 -  It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.
4. Enter the Drill Point parameters:

X Coordinate and Y Coordinate

These are the X and Y coordinates for the drill point.

By Graphic

If this box is checked, the X and Y coordinates will be selected with the mouse.

5. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
6. Click **OK**.
7. If you checked the **By Graphic** box, click with the mouse to place drill points on your design. Right-click to release the tool. If the **By Graphic** box was cleared, a single drill point will be created at the specified coordinates.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.




Creating Drill Circles

Menu: Toolpath / Drill / Drill Circles

Toolbar: Toolpath / Drill / Drill Circles

The Drill Circle strategy creates a circular pattern of drill points, centered at the location specified.

To create a drill circle:

1. Click **Drill** then **Drill circle**. 
2. In the Drill Point dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
3. In the tool listing that appears in the list of cuts at the top of the dialog, set the Depth of the drill holes.
 It is possible to enter fractional values such as 1/2 in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of 3/16 is 0.1875.
4. Enter the Drill Circle parameters:

Circle Diameter

This is the diameter of the circle around which the drill points will be placed. This dimension must be greater than zero to be legal.

Angle	You may want to rotate the drill points on the circle to place them correctly. When the angle is zero the first drill point will be placed on the circle directly to the right of center. You may enter a value between 0 and 360 for this value.
Number of Holes	This specifies the number of drills to be placed. A value of 1 or greater should be entered.
X Coordinate and Y Coordinate	These are the X and Y coordinates for the center of the circle of drill points you are defining.
By Graphic	If this box is checked, the X and Y coordinates will be selected with the mouse.

5. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
6. Click **OK**.
7. If you checked the **By Graphic box**, click with the mouse to place drill circles on your design. Right-click to release the tool. If the By Graphic box was cleared, a single drill circle will be created at the specified coordinates.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template




To load the settings previously saved as a template, select the template from the **Strategy** list.

Creating Drill Arrays


Menu: Toolpath / Drill / Drill Array

Toolbar: Toolpath / Drill / Drill Array

The Drill Array strategy creates a 'matrix' of drill points with a set number of rows and columns, and a specified amount of spacing between them.

3. Click **Drill** then **Drill Array**. 
4. In the Drill Array dialog, select the desired tool from the Available Tools list and click Add Tool.
 -  The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
5. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the drill holes.
 -  It is possible to enter fractional values such as 1/2 in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of 3/16 is 0.1875.

6. Enter the Drill Array parameters:

Columns	The number of vertical drill columns.
Rows	The number of horizontal drill rows.
Spacing	<p>The options that define the spacing of the drill points in the array.</p> <p> All spacing is applied between the center drill points of the columns and rows, not between the outside edges of the drill cuts.</p> <p>Width by overall width If selected, enter the overall width of the array. The drill points will be evenly spaced horizontally within this width.</p> <p>Width by horizontal spacing If selected, enter the amount of horizontal space between the columns of drill points.</p> <p>Height by overall height If selected, enter the overall height of the array. The drill points will be evenly spaced vertically within this height.</p> <p>Height by vertical spacing If selected, enter the amount of vertical space between the rows of drill points.</p>
X Coordinate and Y Coordinate	These are the X and Y coordinates for the center of the array of drill points you are defining.
By Graphic	If this box is checked, the X and Y coordinates will be selected with the mouse.

- 7. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
- 8. Click **OK**.
- 9. If you checked the **By Graphic** box, click with the mouse to place drill arrays on your design. Right-click to release the tool. If the **By Graphic** box was cleared, a single drill circle will be created at the specified coordinates.

Saving Strategy Settings as a Template

To save the current settings as a template:

- 1. Click the **Save as** button.
- 2. Type a name for the template and click **OK**.


Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.




Creating Drill Points in the Centers of Contours

Menu: Toolpath / Drill / Drill Points in Center

Toolbar: Toolpath / Drill / Drill Points in Center

 The Drill Centers strategy can only be used with closed contours.

The Drill Centers strategy creates drill points at the centers of each of the selected contours. You can specify to select only circles and a range of sizes to use, if desired.

1. Select the contours you want to create drill points in the center of.
2. Click **Drill** then **Drill centers.** 
3. In the Drill Centers dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 -  The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the drill hole.
 -  It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.
5. Select the Drill Centers parameters:

All contours	If selected, holes will be drilled in the centers of all the selected contours.
Circles only	If selected, holes will only be drilled in the centers of the selected circles. Selecting this option displays the following additional options.
All circles	If selected, holes will be drilled in the centers of all selected circles.
By diameter	If selected, holes will be drilled in the centers of all selected circles whose diameters are within the Tolerance of the Check Diameter specified.
6. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
7. Click **OK**.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template




To load the settings previously saved as a template, select the template from the **Strategy** list.




Creating Drill Points Along a Contour

Menu: Toolpath / Drill / Drill Contour

Toolbar: Toolpath / Drill / Drill Contour

The Drill Contours strategy creates drill points along selected contours, based on the number and spacing specified.

- 1. Select the contours you want to create drill points in the center of.
- 2. Click **Drill** then **Drill contours**. 
- 3. In the Drill Contours dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 -  The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
- 4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the drill hole.
 -  It is possible to enter fractional values such as 1/2 in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of 3/16 is 0.1875.
- 5. Select the Drill Contours parameters:

Spacing Type	Select the way the drill points will be placed along the contours you have selected:
Hold Number	A specified number of drill points will be spaced evenly along the length of the contours. The number is specified by the Number of Holes parameter.
Hold Spacing	Drill points will be spaced along the length of the contours at a distance specified by the Spacing parameter.
Corners	Drill points will be created at all corners in the contours. <ul style="list-style-type: none"> Circles and ellipses have no corners, and will have no drill points created for them.
Inside Corners	Drill points will be created at all inside corners in the contours. <ul style="list-style-type: none"> Circles and ellipses have no corners, and will have no drill points created for them.
Outside Corners	Drill points will be created at all outside corners in the contours. <ul style="list-style-type: none"> Circles and ellipses have no corners, and will have no drill points created for them.
Number of Holes	The number of holes used with the Hold Number spacing type.
Spacing	The spacing between drill points used with the Hold Spacing spacing type.

6. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
7. Click **OK**.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.




Creating Drill Points at the Corners of the Plate

Menu: Toolpath / Drill / Drill Corners

Toolbar: Toolpath / Drill / Drill Corners

 The Drill Corners strategy is available only if a plate has been defined.

The Drill Corners strategy creates drill points in the corners of the plate, with the ability for additional drills in between each corner.

1. Click **Drill** then **Drill Corners**. 
2. In the Drill Corners dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 -  The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
3. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the drill hole.
 -  It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.
4. Enter the Drill Corners parameters:

Inset X	The horizontal inset from the corners of the plate. If it is left at 0.00 the drill points will be centered on the corners of the plate.
Inset Y	The vertical inset from the corners of the plate.
Add Copies X	Enter a number here to create additional drill points spaced evenly between the corner drills in the horizontal axis.
Add Copies Y	Enter a number here to create additional drill points spaced evenly between the corner drills in the vertical axis.
5. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
6. Click **OK**.

7. If you checked the **By Graphic** box, click with the mouse to place drill points on your design. Right-click to release the tool. If the **By Graphic** box was cleared, a single drill point will be created at the specified coordinates.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

Editing Drill Cut Parameters

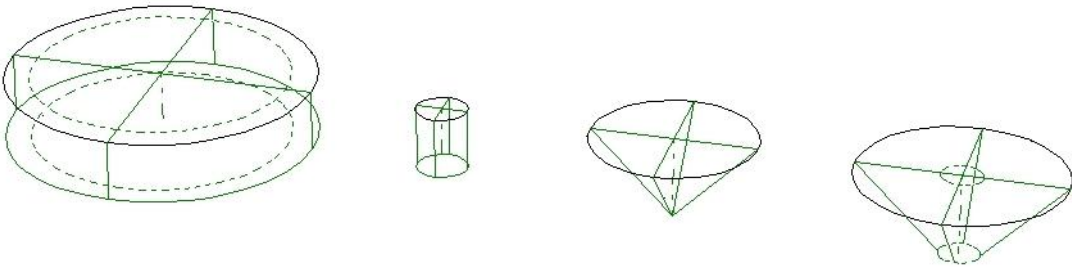
The drill cut parameters are very similar to the cut parameters in the other strategies, with a couple of key differences. They are all the same within the different Drill Strategies. The following sections identify the unique cut parameters for the drill.

Diameter by tool vs. Exact diameter – If the 'Diameter by tool' option is selected, the drill will truly be a drill point, meaning that the size of the hole is based entirely on the diameter of the tool at a given depth.

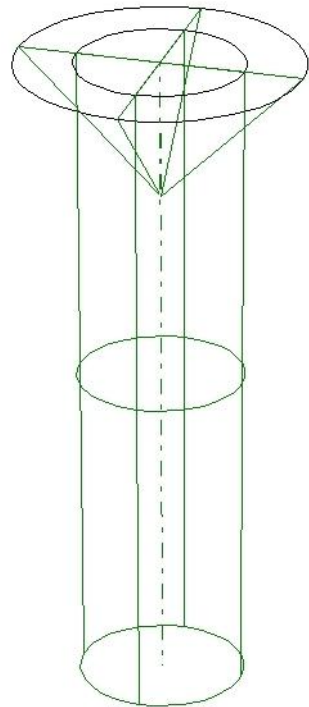
If the 'Exact diameter' option is selected, the user can define a diameter that is larger than the tool's diameter, and the drill point will be created as a circle. The circle will be sized to make the drill hole the diameter specified for the selected tool at the depth defined.

It is important to point out that EnRoute takes into account the shape of the tool in determining the diameter of the tool at a given depth. If you are using a conic tool to create the drill, and specify an exact dimension at a depth that is less than the height of the bevel, EnRoute will automatically take into account the real diameter of the tool at the depth specified.

The following screen image illustrates a number of drills that show drills defined both 'by tool' and 'by diameter'.



Countersink – If a tool is selected that is capable of engraving (conic, engrave, tapered) then if the 'Diameter by tool' option is selected, another option is made available. Countersink lets you specify the diameter of a hole you would like to make that takes into account the bevel on the tool. EnRoute will then automatically calculate how deep the tool should go in order to create a countersink hole with an outer diameter as specified. This is an option designed to aid in designing drill holes that will accommodate screws of a certain size in the finished piece.



Depths, Passes, Feeds and Speeds – These parameters are pretty much just the same in the drills as they are in all of the other strategies. Review the detailed descriptions provided in the Routing Offset Strategy section for more information.

Creating Slot Toolpaths

Menu: Toolpath / Slots

Toolbar: Toolpath / Slots

A slot is a special tool that allows you to define a tool to cut along a line, making a slot that is the width of the tool diameter. Like drills, slots can include more than one tool, and more than one depth.



1. Click **Slots**.
2. In the Slot dialog, select the desired tool from the **Available Tools** list and click **Add Tool**.
 - ☰ The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.
3. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the drill hole.
 - ☰ It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.
4. Enter the Drill Corners parameters:

Start X and Y Coordinates

The X and Y coordinates of the starting point of the slot.

End X and Y Coordinates

The X and Y coordinates of the endpoint of the slot. These must be different than the coordinates of the starting point.

By graphic

If this box is checked, the coordinates of the starting and end points will be selected with the mouse.

Back and forth

If checked, the tool will move from the start point to the end point and return to the starting point before being lifted from the plate.

5. Add additional drill cuts using other tools as necessary. All cuts will share the same drill point coordinates.
6. Click **OK**.
7. If you checked the **By Graphic** box, click with the mouse to place the starting and end points on your design. Right-click to release the tool. If the **By Graphic** box was cleared, the slot will be created between the specified coordinates.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

Creating an Engrave Toolpath


Menu: Toolpath / Engrave


Toolbar: Toolpath / Engrave


Engraving is a traditional machining function that has many applications. In one sense it is the most straightforward type of toolpath creation, and in another sense it is the most complex.


Two-dimensional engraving is essentially just the creation of toolpaths that follow the contours of the contours at a specified depth. 3D toolpaths offer better results: they are able to cut out sharp corners better when using conic tools, for instance. However, 3D toolpaths using conic tools require a tremendous number of computations in order to accurately and reliably create the desired effect.


For the user it is a simple matter of deciding if the toolpaths should be 2D or 3D. The software then performs the required computations to create the appropriate type of toolpaths.

 If you are defining a deep engrave strategy it is often better to use a fill strategy with a 3D Clean pass so that if the engrave toolpath doesn't go all the way to the bottom, the fill tool can finish milling the bottom surface.


1. Select the contour you want the toolpath to follow.
2. Click **Engrave**. 
3. In the Engrave dialog, select the desired tool from the **Available Tools** list. Click **Add Tool**.

 The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.

 Only those tools that are capable of engraving, including conic, engrave and tapered tools, may be defined as engrave cuts.

 The first tool in the list is always defined as the Engrave tool, and it is typically used as the main cutting tool. Any additional tools serve as Rough tools, primarily used to remove excess material. The typical scenario in the case of a deep engraving toolpath is to define a Rough cut that uses an end mill tool.


4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the cut.

 It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.

5. Add additional cuts using other tools as necessary.
6. Set the Engrave parameters:

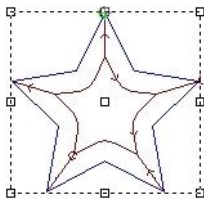
3D Engrave toolpath

Check to create the engrave cut as a 3D engrave toolpath. If cleared, the cut will be a 2D toolpath.

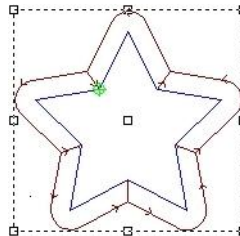
 Remember that only tools capable of 3D engraving, including conic, engrave and tapered tools, may be defined as 3D engrave toolpaths.

External or Internal


If the 3D option is selected for the engrave toolpath, it is necessary to define whether the toolpaths should pass to the inside or the outside of the contours in the selection. Most typically, the engraving toolpaths will be defined on the inside of the contours. The following images show 3D toolpaths created on the inside and outside of a contour:




Internal



External

 Because external 3D toolpaths require all objects in the selection to be combined into one toolpath group, they are not compatible with other types of toolpaths in a contour. If existing toolpath groups are included in the selection, their toolpaths will be deleted when these toolpaths are created.

7. Click **OK**.

 Once you create the toolpath, the contour will become part of the toolpath group.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

Engrave Cut Parameters

The engrave cut parameters include the standard categories of Depths, Passes, Feeds and Speeds, and Direction. The first cut tool is automatically defined as the Engrave tool, and if a second tool is defined, it is classified as the Rough tool. Toolpaths for the Rough tool are only created if required by the Engrave cut.


Creating Pyramid Toolpaths

Menu: Toolpath / Pyramid

Toolbar: Toolpath / Pyramid

The Pyramid Strategy allows you to create a beveled type of artwork that is also commonly referred to as chiseled or sculpted artwork. The Pyramid Cut utilizes a beveled tool, most commonly a conic tool, to create the beveled effect. This cut is unique in that it is necessary for the software to calculate the depth of cut based on the selected tool, and the selected contours. The cut depth is based on the widest 'stroke' in the design. The software then places toolpaths at the proper depth based on the maximum cut depth it calculates.


The finished result from this strategy can be very striking, and it can be effectively utilized on very small pieces all the way up to designs that are several feet tall and wide.


 Remember to check the Pyramid parameter for 'Move to bottom of plate' in order for the Return Height to be created correctly.

1. Select the contour you want the toolpath to follow.


2. Click **Pyramid**. 

3. In the Pyramid dialog, select the desired tool for the pyramid cut from the **Available Tools** list and click **Add Tool**.


 Only those tools that are capable of engraving, including conic, engrave and tapered tools may be defined as pyramid cuts.

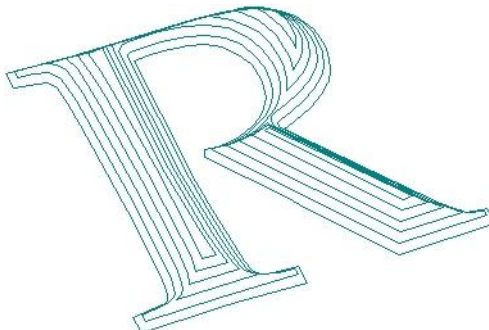
 The **Sort tools by** list determines the parameter that is used to sort the Available Tools list. It can also filter the tools that are displayed so that only a certain type of tool is visible.

4. In the tool listing that appears in the list of cuts at the top of the dialog, set the **Depth** of the cut.

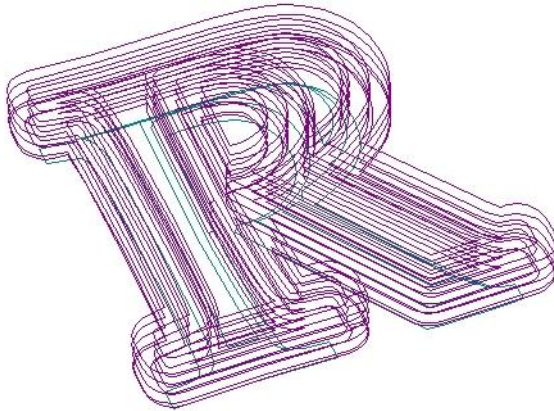
 It is possible to enter fractional values such as $\frac{1}{2}$ in order to let the program calculate the decimal equivalent. That way it isn't necessary to remember that the decimal equivalent of $\frac{3}{16}$ is 0.1875.

5. Add additional cuts using other tools as necessary.

 The first tool in the list is always defined as the Pyramid tool, and it is typically used as the main cutting tool.



- 📄 The second tool is defined as the Rough tool, and is primarily used to remove excess material. The Rough tool is typically a medium-sized end mill tool.

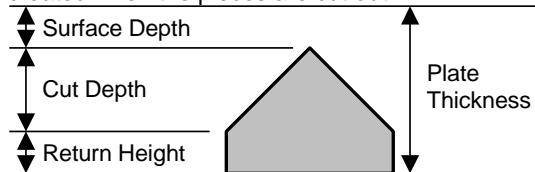


- 📄 The third tool is defined as the Offset tool, and is used to cut around the outside of the Pyramid cut, either to create a relief, or to cut the pyramid away from the surrounding plate.

6. Set the Pyramid parameters:

Return Height

The height of any vertical 'edge' that is to be created below the beveled portion of the contour. In order for the return height to be meaningful, it is necessary to move the toolpath down to the bottom of the material so that the return will be created when the pieces are cut out.



Move to bottom of plate

If checked, the bottom edge of the return height (if any) will be aligned with the bottom of the plate.

If this option is cleared, the toolpaths will be placed such that the top of the bevel will be at the surface of the material.

7. Click **OK**.

- 📄 Once you create the toolpath, the contour will be part of the toolpath group.

Saving Strategy Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

Loading Strategy Settings from a Template

To load the settings previously saved as a template, select the template from the **Strategy** list.

Pyramid Cut Parameters

The cut parameters in this strategy are much the same as the other strategies. The key difference is that for the most part the software will automatically define the depths of the cuts for you based on the selections you make at the strategy level. It is possible to override the Surface Depth for the cuts, although this is not typically recommended.

You may at times want to increase the number of passes for a given cut in order to take into account material densities. By default, EnRoute will utilize 90 percent of the tool's cutting depth on the Pyramid cut. For soft material this is likely appropriate, but for denser material such as wood, you will likely want to decrease the pass depths by increasing the number of passes.

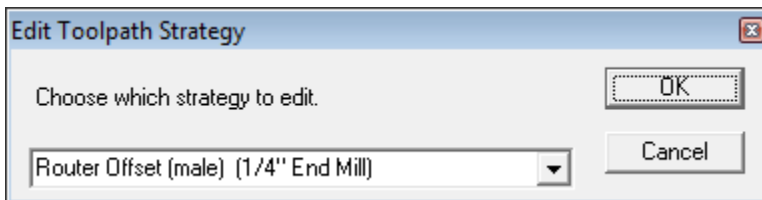
Editing Toolpaths

Menu: Toolpath / Edit Toolpaths

After toolpaths are created, the software maintains all of the information about how the toolpaths were created. You can edit the toolpaths at any time and change the cuts and parameters that define the toolpath. When you are finished, the software regenerates the toolpath using the new settings.

To edit a toolpath:

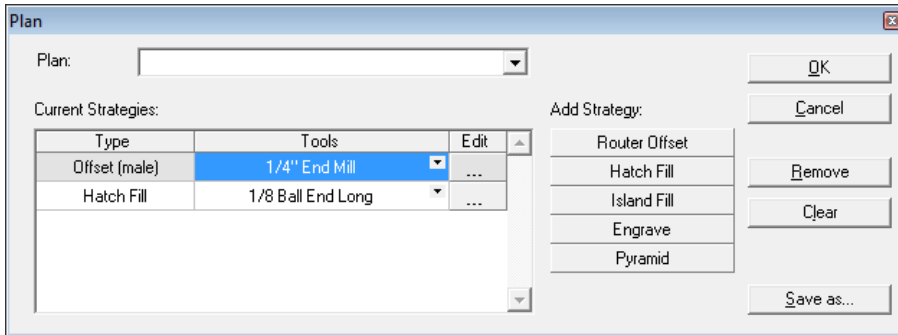
1. Select the toolpath.
2. From the **Toolpath** menu, select **Edit toolpaths**.
3. If the toolpath group contains more than one strategy, you will be prompted to choose which strategy to edit. Select the strategy you want to edit and click **OK**.



4. In the dialog for the strategy, make the desired changes to the settings and click **OK**.

Editing the Toolpath Plan

The *plan* is the set of all strategies contained in a toolpath group. Editing it allows you to edit the individual strategies, delete them, add new strategies, and save the entire plan as a template.



Accessing the Plan

To access the plan:

1. Select the toolpath.
2. From the **Toolpath** menu, select **Edit toolpaths**.
3. If necessary, select a strategy and click **OK**.
4. In the strategy dialog, click **Edit Plan**.

Editing Strategies

To edit a strategy:

1. Select the strategy in the strategy list and click on the ellipses in the **Edit** column.
2. Edit the strategy in the strategy dialog.
3. Click **OK**.

Adding Strategies

To add a strategy to the plan:

1. Click on the appropriate button under **Add Strategy**.
2. Create the strategy using the strategy dialog.
3. Click **OK**.

Removing Strategies

To delete a strategy from the plan:

1. Select the strategy in the strategy list.
2. Click **Remove**.

Clearing All Strategies

To delete all the strategies from the plan, click **Clear**.

Saving Plan Settings as a Template

To save the current settings as a template:

1. Click the **Save as** button.
2. Type a name for the template and click **OK**.

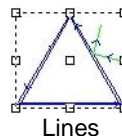
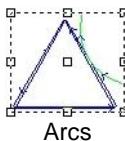
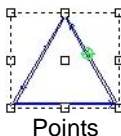
Loading Plan Settings from a Template

To load the settings previously saved as a template, select the template from the **Plan** list.

Editing Entry/Exit Points

Each cut in a toolpath has its own entry and exit points. These are the points at which the tool will enter and exit the plate material as it makes the cut. Often the entry and exit points are at the same point.

By default, the entry and exit points are drill points, indicating that the tool will drill straight down into the material at the start point and lift straight out at the end point. The entry and exit points may also be arcs or lines, however, in which case the cutting tool will move laterally into the material on the line or arc specified and then begin cutting along the toolpath. If a 3D line or arc is specified, the tool will move down into the material on a diagonal line as viewed from the side.



☞ Not all toolpath start points may be moved, and not all start points are eligible to include Entry/Exit toolpaths. Generally this type of editing is most applicable to Routing Offset toolpaths.

Moving Entry/Exit Points

Menu: Toolpath / Edit Entry/Exit

To move an entry/exit point:


1. Select the toolpaths you want to edit.
2. From the **Toolpath** menu, select **Edit Entry/Exit**.
3. Click on the entry/exit point and drag it to its new location along the toolpath.

☞ Moving entry/exit points on a toolpath can use the snap settings to more precisely locate the desired point.

Moving Entry/Exit Points for Individual Cutting Passes

By default, each of the cutting passes for a given cut will have identical entry/exit points. When you move the entry exit points for one, you move them for all of the cutting passes.

To move the entry/exit points for a single cutting pass:

1. Press **F10**.
 2. In the View Setup tab of the Setup Preferences dialog, set **Depth** to the cutting depth of the pass you want to edit.
 3. Click **OK**.
-  Only the single cutting pass at this depth is displayed/editable.
4. Select the toolpath you want to edit.
 5. From the **Toolpath** menu, select **Edit Entry/Exit**.
 6. Click on the entry/exit point and drag it to its new location along the toolpath.
 7. Press **F10**.
 8. In the View Setup tab of the Setup Preferences dialog, set **Depth** back to **All depths**.
 9. Click **OK**.

Editing Entry/Exit Cut Parameters

Menu: Toolpath / Edit Entry/Exit

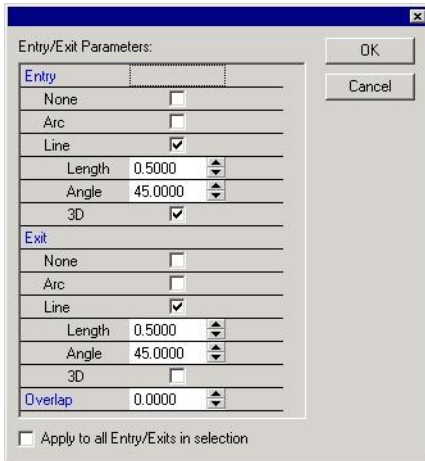
In order to change the type of entry/exit point in use, or to make them 3D, it is necessary to edit the entry/exit parameters for a cut.

The entry/exit parameters can be edited along with the other cut parameters for the cut, or they can be edited separately.

For details on editing all cut parameters, see “Editing Cut Parameters”.

To edit only the entry/edit points for a toolpath:

1. Select the toolpaths you want to edit.
2. From the **Toolpath** menu, select **Edit Entry/Exit**.
3. Click on the entry/edit point you want to edit.
4. Right-click and select **Edit Entry/Exit** from the shortcut menu.

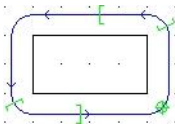


5. In the Edit Entry/Exit dialog that displays, edit the entry/exit parameters for the strategy.
6. Check **Apply to all Entry/Exit points in selection** to apply the changes to all the entry/exit points in the selected toolpaths.
7. Click **OK**.

Editing Bridges

The Edit Bridges tool allows you to edit individual bridges, moving them and changing their parameters. It is also possible to insert and delete bridges, and globally change the parameters of all the bridges in the selection.

Bridges are small tabs of material that are left connecting parts which would otherwise be completely severed from the rest of the design. After the job has finished cutting, the operator can then manually cut the bridges to remove the part from the rest of the plate.



Bridges are displayed as square brackets that ride the line of the toolpath. The area in between the brackets will form the bridge.

Moving Bridges

To move the endpoints of a bridge:

1. Select the toolpath you want to edit.
2. From the **Toolpath** menu, select **Edit Bridges**.
3. Click and drag the endpoints of the bridges to the desired locations.



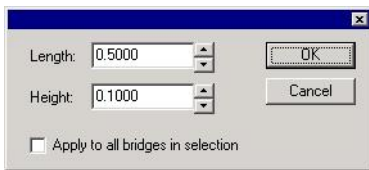
Bridges cannot be moved on top of the entry/exit points or other bridges.

4. Right-click and select **Exit this tool** to release the tool.

Editing Bridge Parameters

To edit the parameters of a bridge:

1. Select the toolpaths you want to edit.
2. From the **Toolpath** menu, select **Edit Bridges**.
3. Right-click and select **Edit Bridge** from the shortcut menu.



4. Set the **Length** and **Height** of the bridges to the desired values.
5. Check **Apply to all bridges in selection** to apply the changes to all the bridges in the selected toolpaths.
6. Click **OK**.
7. Right-click and select **Exit this tool** to release the tool.

Inserting Bridges

To insert a bridge:

1. Select the toolpaths you want to edit.
2. From the **Toolpath** menu, select **Edit Bridges**.
3. Right-click and select **Insert Bridges** from the shortcut menu.
4. Click on the toolpaths to insert bridges using the current bridge parameters.
5. Right-click and select **Exit this tool** to exit the **Insert Bridges** tool.
6. Right-click and select **Exit this tool** to exit the **Edit Bridges** tool.

Deleting Bridges

To delete a bridge:

1. Select the toolpaths you want to edit.
2. From the **Toolpath** menu, select **Edit Bridges**.
3. Right-click on a bridge and select **Delete Bridge** from the shortcut menu.
4. Right-click and select **Exit this tool** to exit the tool.

Deleting Toolpaths

Menu: Toolpath / Delete Toolpaths

Toolbar: Toolpath / Delete Toolpaths

The Delete Toolpaths tool allows you to delete all the toolpaths in a toolpath group. The original contours in the toolpath group will remain.

To delete a toolpath:

1. Select the toolpaths you want to delete.

2. Click **Delete toolpaths**. 

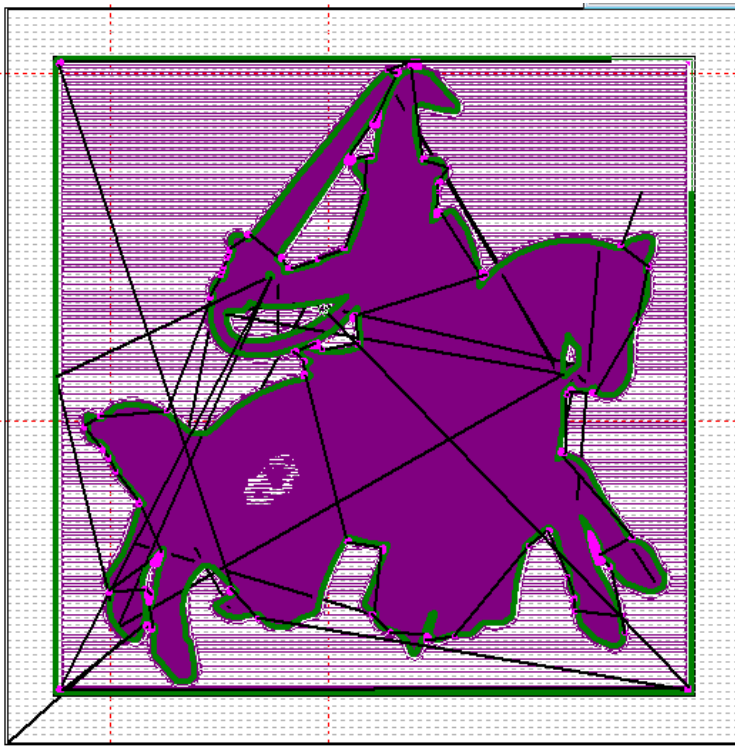
9. Previewing Output

Simulating Output in 2D

Menu: Machining / Simulate 2D

Toolbar: Output / Simulate 2D

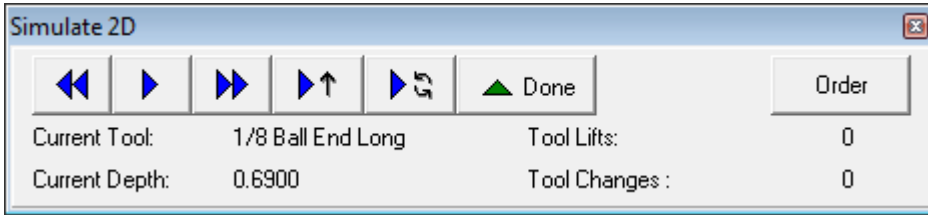
The Simulate 2D tool allows you to preview the output of your toolpaths in the order they will be sent to the machine. This allows you to review the output before cutting it to make sure that the toolpaths and their order are correct.




To simulate output of your design in 2D:








1. Click Simulate 2D.






2. In the Simulate 2D dialog, use the controls provided to play the simulation. The controls are similar to those on a CD player or tape recorder:

 Hit the up and down arrow keys to speed up and slow down the simulation.

	Rewind	This button will rewind the simulation to the beginning.
	Draw next segment	This will advance the simulation until the next segment has been output.
	Play	This will start the simulation and allow it to play until the end.
	Next Lift	This will advance the simulation to the next tool lift.
	Next Tool	This will advance the simulation to the next tool change.
	Done	This will end the simulation and close the Simulate 2D dialog.
	Order	This button displays a Toolpath Ordering dialog that is identical to the one in the Output dialog. Changes in Priority Ordering, Tool, Strategy and Sort Method may all be accomplished directly in the 2D Simulation tool so that the effect of any change may be viewed.

 After any ordering changes are made, be sure to click on the 'Update Order' button so that the changes are reflected in the output and simulation.

3. Click **Done** to end the simulation.

Simulating Output Using an Orthographic View

Menu: Machining / Simulate Ortho

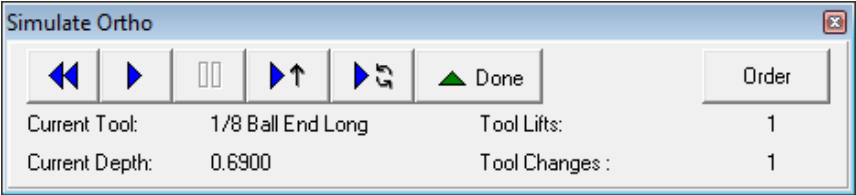
Toolbar: Machining / Simulate Ortho


The Simulate Ortho tool displays an animated orthographic view of the path each tool will take and the order in which it will cut. If you see that you have made an error in the toolpath or the order of the cut, you have the opportunity to make corrections.



To simulate output of your design using an orthographic view:

1. Click **Simulate Ortho**. 



2. In the Simulate Ortho dialog, use the controls provided to play the simulation. The controls are similar to those on a CD player or tape recorder:
 Hit the up and down arrow keys to speed up and slow down the simulation.



Rewind This button will rewind the simulation to the beginning.



Draw next segment This will advance the simulation until the next segment has been output.



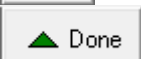
Pause This will stop the simulation.



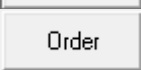
Next Lift This will advance the simulation to the next tool lift.



Next Tool This will advance the simulation to the next tool change.



Done This will end the simulation and close the Simulate 2D dialog.



Order This button displays a Toolpath Ordering dialog that is identical to the one in the Output dialog.

Changes in Priority Ordering, Tool, Strategy and Sort Method may all be accomplished directly in the 2D Simulation tool so that the effect of any change may be viewed.

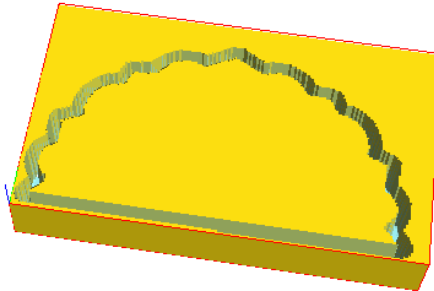
- After any ordering changes are made, be sure to click on the 'Update Order' button so that the changes are reflected in the output and simulation.
- Click **Done** to end the simulation.

Create Rendered View of Toolpaths


Menu: Machining / Simulate 3D

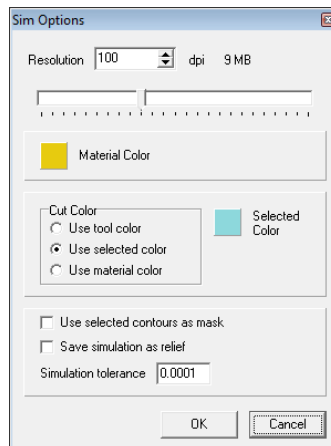
Toolbar: Machining / Simulate 3D

This tool displays a rendered solid model view of the results of the toolpaths cutting the material. It accurately renders all of the different tool shapes, so you can see the results of both 2D and 3D toolpaths.



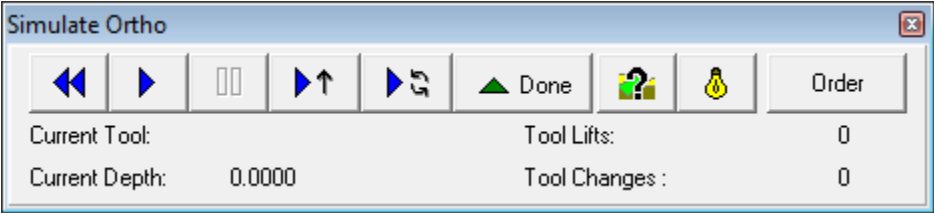
To simulate output of your design using a rendered view:

- Click **Create Rendered Simulation** of Toolpaths.  This will open the Simulate Options Dialog.
- Enter the desired parameters below and then click **OK**.









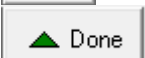
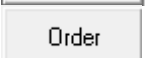
- Resolution** – Image resolution refers to the spacing of pixels in the image and is measured in pixels per inch. The higher the resolution, the more pixels in the image. Higher resolutions allow for more detail but will take longer to display.

- **Material Color** – The material color is the color of the relief. You can change this color by clicking on the material color icon and choosing the color from the color chart.
- **Cut Color** – The cut color has three choices:
 - ✧ Use tool color – The simulation will show using the colors assigned to each tool type in the preferences section.
 - ✧ Use Selected color – The selected color is shown to the right. This color can be changed by clicking on the icon and selecting a new color from the chart.
 - ✧ Use Material color – Select this parameter if you would like the simulation to be shown using the material color.
- **Use selected contours as mask** – Check this to use the selected contours as a mask for the area that you want to simulate.
- **Save simulation as relief** – The simulation will be saved as a relief. The relief will be placed in the plate.
- **Simulation tolerance** – The tolerance controls how closely the simulation will display to the relief toolpaths.



3. In the **Create Rendered Simulation** dialog, use the controls provided to play the simulation.

 Hit the up and down arrow keys to speed up and slow down the simulation.

	Rewind	This button will rewind the simulation to the beginning.
	Draw next segment	This will advance the simulation until the next segment has been output.
	Pause	This will start the simulation and allow it to play until the end.
	Next Lift	This will advance the simulation to the next tool lift.
	Next Tool	This will advance the simulation to the next tool change.
	Done	This will end the simulation and close the Simulate 2D dialog.
	Order	This button displays a Toolpath Ordering dialog that is identical to the one in the Output dialog.
<p>Changes in Priority Ordering, Tool, Strategy and Sort Method may all be accomplished directly in the 2D Simulation tool so that the effect of any change may be viewed.</p>		


- After any ordering changes are made, be sure to click on the 'Update Order' button so that the changes are reflected in the output and simulation.
- 4. Click **Done** to end the simulation.
- Be sure to activate the Rendered View icon .

Image with rendered view activated.

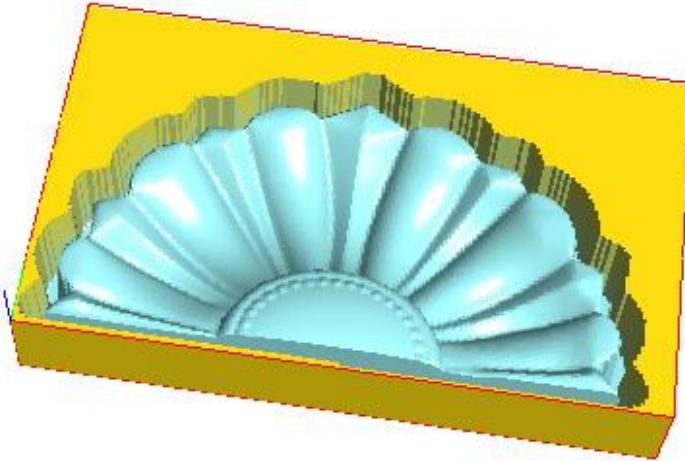
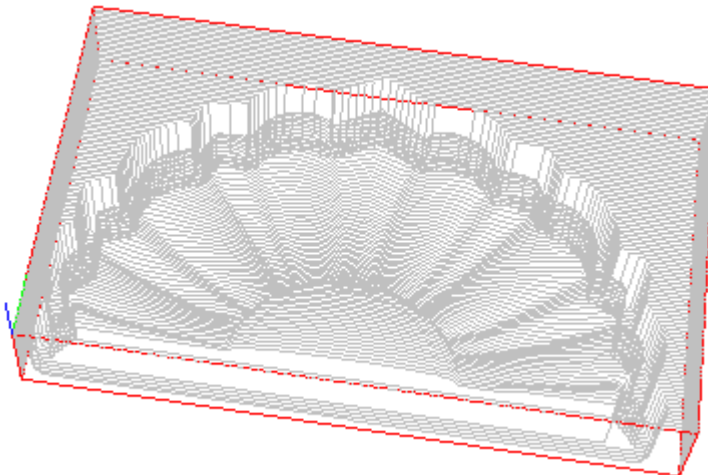


Image without rendered view activated.



10. Configuring the Software

Configuring Machine Drivers

Menu: Setup / Machine Setup

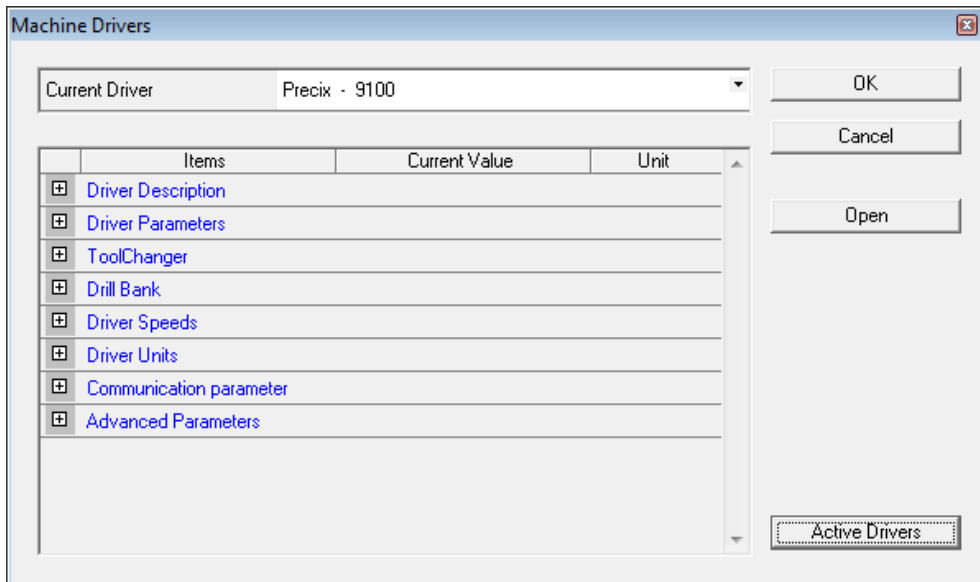
Toolbar: Libraries / Driver Setup

The Machine Drivers dialog is used to select the appropriate driver for your machine, and to configure it, if necessary, to adjust specific parameters. The drivers will typically be located in a folder called **NDrivers**, located in your program folder. The driver configuration files have a **CNF** extension, along with a driver file with the same name, with either a **PST** or **DLL** extension. Any changes you make to the driver configuration parameters are saved in the **CNF** file.

After a driver for your machine is configured, it is not typically necessary to make changes to the driver configuration unless something changes. The exception to this may be changes to the tools loaded in your machine's tool changer if it is configured with one.

To set up a machine:

1. Click **Driver Setup**.



2. Select the driver for the machine from the **Current Driver** list.

The **Current Driver** list displays the drivers which have been selected as active. If the driver for your machine is not in the list, see "Selecting Active Drivers" for instructions on adding the driver to the list.

3. Edit the driver parameters.

Driver Description

Name	The name of the driver file being edited.
Model	The model name of the driver being edited.
Description	The description provides a brief description of the driver, such as what type of driver it is, or specifically what model of machine it has been created for.



Driver Parameters

Width	This provides the width dimension for the machine (typically the X-axis of the machine).
Height	This provides the height dimension for the machine (typically the Y-axis of the machine).
Z Lift	This gives the maximum Z-axis dimension, or thickness, of the machine.
Home Position	This section defines what type of home definition is being provided, including User Defined , Plate , Design , and Selection . It is most typically defined as User Defined , which provides a specific location that is not dependant on elements of the design. The coordinates of the Home Position can also be defined, along with where that corresponds to a location on the machine. After being originally specified in the driver, these values are not typically changed.

Tool Changer	<p>The Tool Changer section specifies whether your machine has a tool changer available, how many tools are present, and which specific tools are available in the turret.</p> <p>See “Configuring a Tool Changer” for details on setting up a tool changer.</p>
---------------------	--

Drill Bank	<p>The Drill Bank section specifies whether your machine includes a Drill Bank. A drill bank is a separate head that provides an array of drill tools to drill a set of holes using a pre-configured spacing. A typical application for a drill bank is to bore holes for shelf mounts as part of the manufacture of cabinet parts.</p> <p>See “Configuring a Drill Bank” for details on setting up a drill bank.</p>
-------------------	---

Driver Speeds	<p>This category contains a definition of the maximum and minimum speeds that are possible with the current machine. They are based on the physical capabilities of the machine, as well as practical values. This will help ensure that you don’t attempt to define speed values that are not possible, or might damage the machine.</p> <p>The units used in this category are based on the units that are specified for the machine. Units are specified by machine manufacturers based on the values that are used by their controllers, so it is not recommended to change the speeds unless you know for sure that the change is appropriate.</p>
----------------------	---

Spindle Speed	This provides a default spindle speed that is used if no spindle speed is provided in the output file.
Feed	These are the feed rates that the machine moves in the X and Y-axes. There are values specified for both 'Within the material' and 'Above the material'. The first refers to cut speeds while the tool is actually cutting material, and the second refers to feed rates as the tool is moving between toolpaths after it has lifted out of the material.
Plunge	These are the speeds that the machine moves in the Z-axis, either as it is moving down into the cut, or moving up out of the cut.
Dwell	It is sometimes desirable for the tool to remain in one position for a short period of time after it has plunged to a depth. This allows the tool to remove chips properly before it begins its cut.
Driver Units	<p>These are the units that are expected by the machine controller for any values it receives. As described above, these values are most often defined by the machine manufacturer and should not be changed by users unless you know for sure that the change is appropriate.</p> <p>The values are specified for Length, Speed and Time. If you make changes to these values and then decide that the changes are not correct, you can click on the 'Use defaults' check box to return the values to their original values.</p>
Communication Parameters	<p>The communication parameters are used when the software is used to send output directly to a machine. These values are configured to match the parameters specified for communicating with the machine.</p> <p>These parameters specify how information regarding distance and angles is converted into machine movements.</p> <p> Please note that many machine controllers are not designed to receive files directly from the software. If you are not sure how to communicate directly with your machine, check with the manufacturer to find out what communication parameters are appropriate, and whether it is possible to send files using this method.</p> <p> Do Not Change these values unless you know for sure that the changes are correct. If the values are specified incorrectly it will make it so that output will be misinterpreted so that distances and speeds will not be correct.</p>

4. Click **OK**.

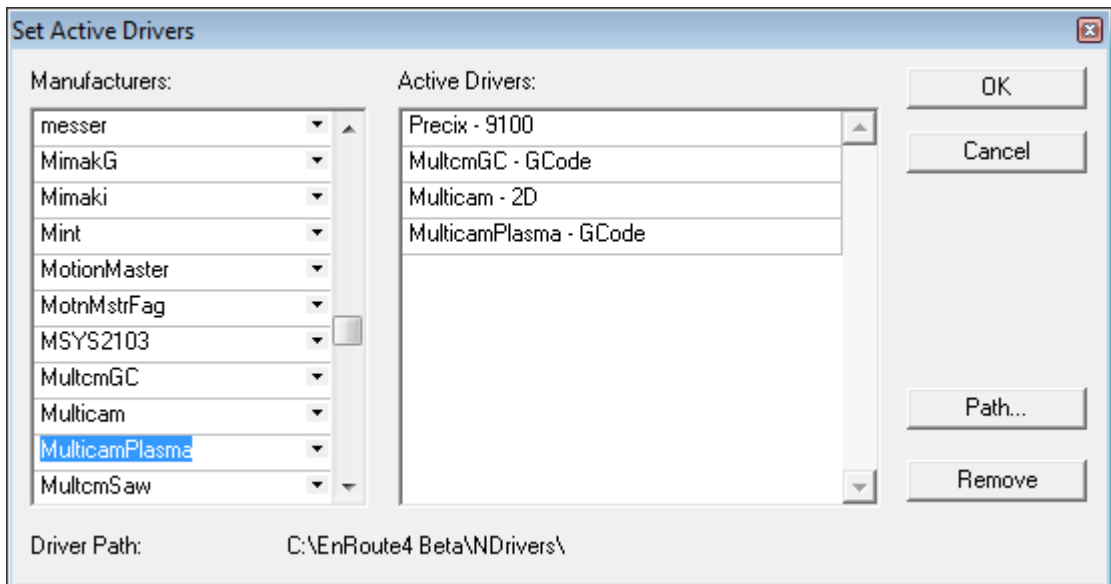
Selecting Active Drivers

The list of active drivers in the **Current Driver** list should list all of the machines commonly used with the software at your location.

Adding an Active Driver

To add a machine to the list of active drivers:

1. Click on the **Active Drivers** button at the bottom of the Machine Setup dialog.



2. In the Active Drivers dialog, select the manufacturer of the machine from the **Manufacturers** list.
3. Click on the arrow on the right end of the manufacturer listing to open a list of their available machine drivers.
4. Select the machine you want to add to the active drivers list from the list of machines.
5. Click **OK**.

Removing an Active Driver

To remove a driver from the list of active drivers:

1. Click on the **Active Drivers** button at the bottom of the Machine Setup dialog.
2. Select the driver in the **Active Drivers** list.
3. Click the **Remove** button.
4. Click **OK**.

Removing a driver from the **Active Drivers** list does not delete it from the **Manufacturers** list.

Configuring a Tool Changer

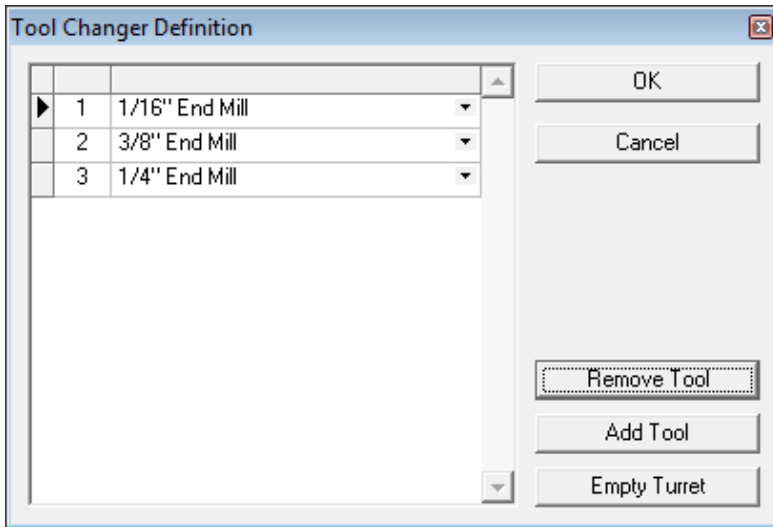
To set up a tool changer:

1. Click the **Auto tool changer** check box in the machine parameters list of the Machine Setup dialog.



The Number of Tools and Edit Tools parameters appear after the Auto Tool changer box is checked.

2. Under **Edit Tools**, click on the **Click to Edit** text to open the Tool Changer Definition window:



3. To add a tool to the list of tools in the turret:
4. Select the tool that is in the position you want the new tool to occupy.
5. Click **Add Tool**. The new tool is added to the list in the selected location, and the existing tools are moved down to make room for it.
6. Using the pull-down list for the new tool, select the type of tool that is loaded in that position in the turret.
7. To edit a tool, select a new type of tool from the pull-down list.
8. To move a tool, click twice on the gray row indicator to the left of the tool listing, and then drag it to the new position in the list. The numbering of the tools will be adjusted to reflect the new position.
9. To remove a tool from the list, select that tool and click **Remove Tool**.
10. To remove all of the tools from the list, click **Empty Turret**.
11. When you have completed editing the Tool Changer Definition dialog, click **OK** to save the changes, or **Cancel** to discard your changes and go back to the Machine Drivers dialog.

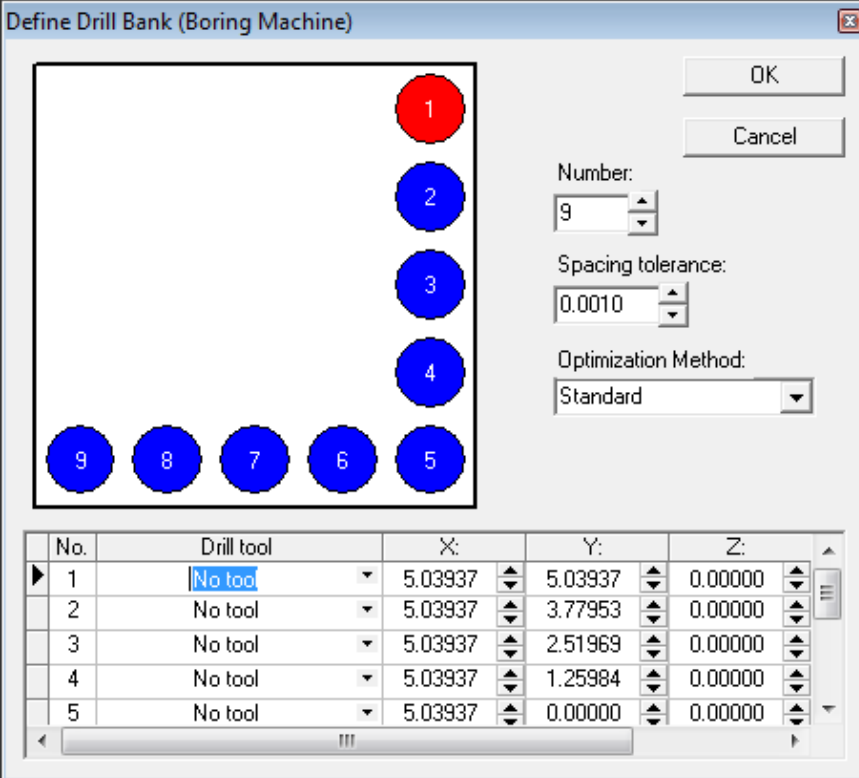


Any changes you made to the number of tools in the turret will be reflected in the **Number of tools** parameter.

Configuring a Drill Bank

To configure a drill bank:

1. Check the **Has drill bank** parameter in the machine parameters list in the Machine Setup dialog.
2. Click on **Click to Edit** to open the Define Drill Bank dialog.




The dialog box titled "Define Drill Bank (Boring Machine)" contains a grid of 9 circular tool slots. Slot 1 is red and contains the number 1. Slots 2 through 5 are blue and contain the numbers 2, 3, 4, and 5 respectively. Slots 6 through 9 are blue and contain the numbers 6, 7, 8, and 9 respectively. To the right of the grid are input fields for "Number" (set to 9), "Spacing tolerance" (set to 0.0010), and "Optimization Method" (set to Standard). At the bottom is a table with 5 rows and 6 columns.

No.	Drill tool	X:	Y:	Z:	
1	No tool	5.03937	5.03937	0.00000	
2	No tool	5.03937	3.77953	0.00000	
3	No tool	5.03937	2.51969	0.00000	
4	No tool	5.03937	1.25984	0.00000	
5	No tool	5.03937	0.00000	0.00000	

3. Set **Number** to the number of drill tools that are present on your machine.
4. For each drill tool in your drill bank:
5. Select the type of Drill Tool that is loaded into that position from the pull-down list provided.
 - 📖 If a tool that is loaded into your machine is not present in the pull-down list, it will need to be entered into the Tool Library before you can set up the machine properly. See "Editing the Tool Library", p. 182 for details.
6. Define the X, Y, and Z location of the tool.
 - 📖 This location is based on the configuration of the machine you are using. It may be necessary to obtain this information from the manufacturer of your machine. This configuration is used to create the final output, so it is very important that the orientation and position of the tools is defined correctly.

7. After completing the configuration of the drill bank, click on **OK** to close this dialog. When you click on **OK** to exit the Machine Drivers dialog, the drill bank configuration you defined will be saved in the driver configuration file.

After you define the layout of your drill bank, you can disable the drill bank in the driver without deleting the layout. Then when you enable the drill bank again, the parameters you defined before will still be present.


 It is important to verify that the driver you are using is capable of using a drill bank. Not all the drivers have been modified to enable them to issue the correct output commands. If you are unsure, please check first with the manufacturer of your machine and then with SA International technical support if necessary to find out whether your driver is capable of using drill banks.

Editing the Material Library

Menu: Setup / Material Library

Toolbar: Libraries / Material Library

The **Material Library** command is used to define the cut speeds for the different types of materials (wood, metal, etc.) that you are routing or engraving. When you use the **Define Plate** command you can select a Material Type to be associated with the plate. The software will then update the plate information to include the cutting speeds associated with the material you select. Material definitions are written to the **MaterialLibrary.ini** file located in the folder where the software was installed.

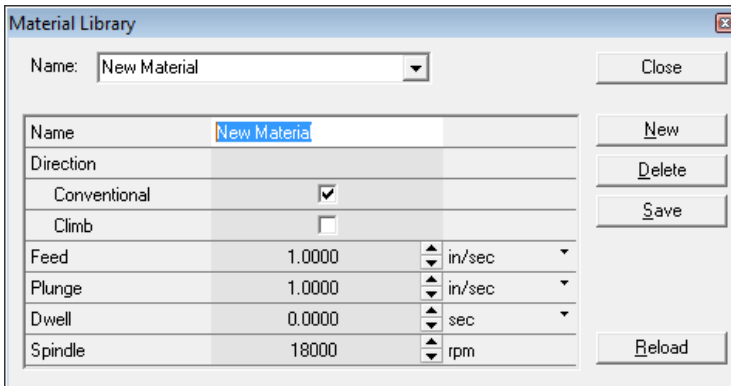
 Some machines and/or drivers may not support the speeds defined in the Material Library. The driver for the machine will override speeds that are defined to be too fast for the machine.

Adding a Material to the Library

Menu: Setup / Material Library

Toolbar: Libraries / Material Library

1. Click Material Library.

A screenshot of the 'Material Library' dialog box. It has a title bar 'Material Library' with a close button. Below the title bar is a 'Name:' label followed by a dropdown menu showing 'New Material'. To the right of this is a 'Close' button. Below the dropdown is a table with columns for 'Name', 'Direction', and 'Speed'. The 'Name' column has 'New Material' selected. The 'Direction' column has 'Conventional' selected with a checked checkbox, and 'Climb' is unselected. The 'Speed' column has values for 'Feed' (1.0000 in/sec), 'Plunge' (1.0000 in/sec), 'Dwell' (0.0000 sec), and 'Spindle' (18000 rpm). To the right of the table are buttons for 'New', 'Delete', 'Save', and 'Reload'.



2. Click **New**.
3. Enter the parameters for the new material.

- | | |
|---------------------|---|
| Name | The name of the material. |
| Direction | Select the direction in which the tool will cut across the plate. |
| Conventional | Left to right milling. |
| Climb | Right to left milling. |
| Feed | The speed at which the cutting tool moves through the material (machining or cutting speed). |
| Plunge | The speed at which the tool moves down into the material. |
| Dwell | The amount of time to wait once the cutting tool has moved down into the material. This is used to make sure the tool is all the way into the material before it starts routing or engraving. It is also useful if there is a concern about chip removal. |
| Spindle | The speed at which the tool will spin. |
4. Click **Save**.

Deleting a Material from the Library

Menu: Setup / Material Library

Toolbar: Libraries / Material Library

1. Click Material Library. 
 2. Select the material from the **Name** list.
 3. Click **Delete**.
-  If you change your mind after deleting a material, click on the **Reload** button to reload the Material Library from the file. The material is not permanently deleted until the Material Library has been saved.
4. Click **Save**.

Editing the Tool Library

Menu: Setup / Tool Library

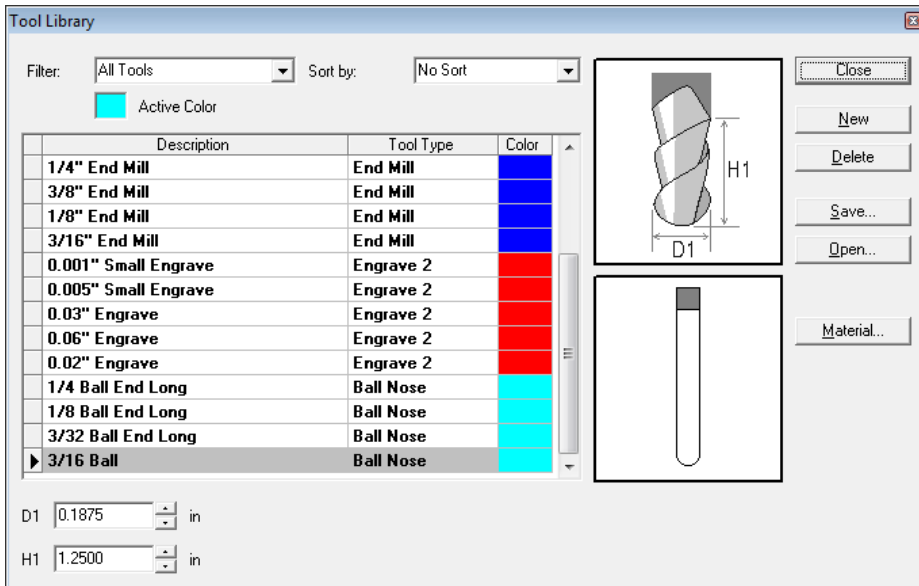
Toolbar: Libraries / Tool Library

The Tool Library stores the list of all cutting tools which have been defined for the software. New tools can be added to the tool library if needed.


Adding a New Tool to the Tool Library

1. Click **Tool library**.






2. **Click on the New button.**
3. **Enter a Tool Name for the new tool.**
4. **Select the Tool Type that best fits the tool being added.**
5. **Click OK.**
6. **Select the newly created tool from the list in the tool library dialog.**
7. **Edit the parameters of the new tool in the fields at the bottom of the tool library dialog.**
8. **Save the changes in the tool library** (see "Saving the Tool Library" below).

 If you are creating tool definitions for tongue and groove joining tools, create them as end mill tools. Give the tongue bit a diameter equal to the outer diameter of the bit, and the groove bit a diameter equal to its inner diameter.

Deleting a Tool from the Tool Library

Menu: Setup / Tool Library

Toolbar: Libraries / Tool Library

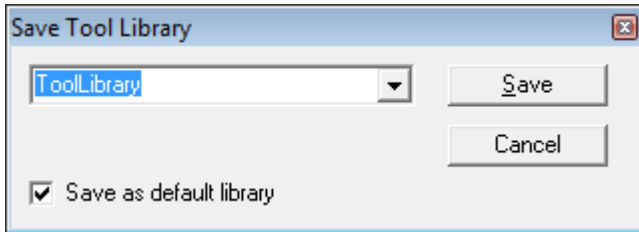
1. Click **Tool library.** 
2. Select the tool you want to delete.
3. Click Delete.
4. Click OK.

Saving the Tool Library

Saving the Default Tool Library

To save the default tool library including any changes:

5. Click **Save**.



6. Check the Save as default tool library box.
7. Click **Save**.


Saving a Custom Tool Library

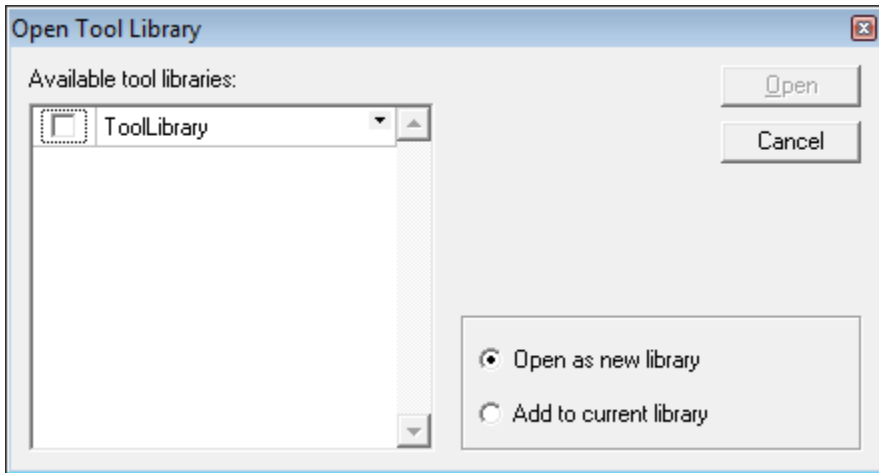
A custom tool library may be designed to fit an individual user or a specific job.

1. **Open** the Tool library.
2. Edit the library so that it contains the tools to be included in this library.
3. Click **Save**.
4. **Enter** the name of the new library to be created.
5. Make sure the **Save as default tool library** box is cleared.
6. Click **Save**.


Opening a Tool Library

Tool libraries are stored by name in the folder the software was installed into. They may be loaded at any time.

1. Click **Tool library**. 
2. Click Open.



3. Place a check mark in the box of the tool library you wish to open.
4. Select one of the following options:
 - Select **Open as a New Library** box is selected; the tools in the selected library will replace the current entries in the tool library.
 - If the **Add to current Library** box is selected, the tools in the selected library will be added to the current entries in the tool library.
5. Click **Open**.

 The default tool library is saved in a file called 'Toollibrary.ini'. This library is automatically opened when EnRoute loads. If you want specific tools to load every time, then save these tools as entries in the default tool library.

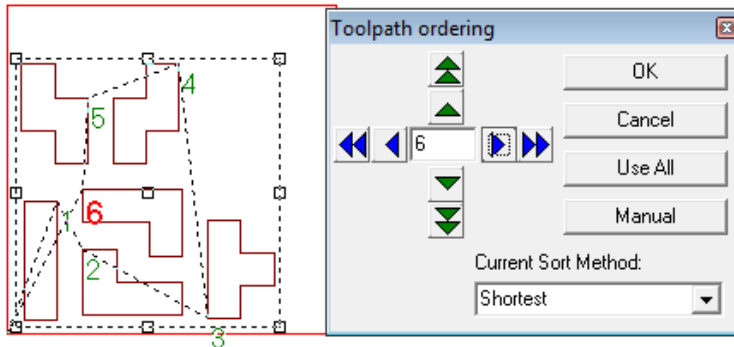
11. Output to Machine

Setting the Toolpath Order

Menu: Machining / Order

Toolbar: Machining / Order

The Toolpath Ordering dialog allows you to review and edit the order assigned to the toolpath groups in your project. The toolpath order influences the order in which cuts will be made when the project is output.



Ordering Toolpath Groups Manually

To manually adjust the order of the toolpath groups:


1. Click **Order**.



2. Use the horizontal blue arrow buttons to select the toolpaths.

 The selected toolpath is indicated with a bold red number.

3. Use the vertical green arrow buttons to change the cut number of the selected toolpath.
4. When finished, click **OK**.

 Clicking the horizontal and vertical arrow keys on your keyboard produces the same results as clicking on the blue & green arrow buttons.

Ordering Toolpaths Using the Mouse

To order the toolpath groups manually using the mouse:


1. Click **Order**.



2. Click **Manual**.
3. Click on each toolpath in the desired order.

Ordering Toolpath Groups Using Sorting Methods

To determine the toolpath order using an automatic sorting method:

4. Click **Order**. 
5. Select the sorting method from the **Current Sort Method** list. The available methods are:
 - Shortest** The toolpath will be sorted to produce the shortest overall set of toolpaths and movements.
 - Rows** The toolpaths will be sorted by ordering them into rows starting from the bottom of the plate. Numbering within rows runs from left to right.
 - Columns** The toolpaths will be sorted by ordering them into columns running from left to right. Numbering within columns runs from bottom to top.
 - Inside Out** The toolpaths will be sorted by position from the center of the plate to the edges.
 - Outside In** The toolpaths will be sorted by position from the edges of the plate to the center.
6. Click **OK**.

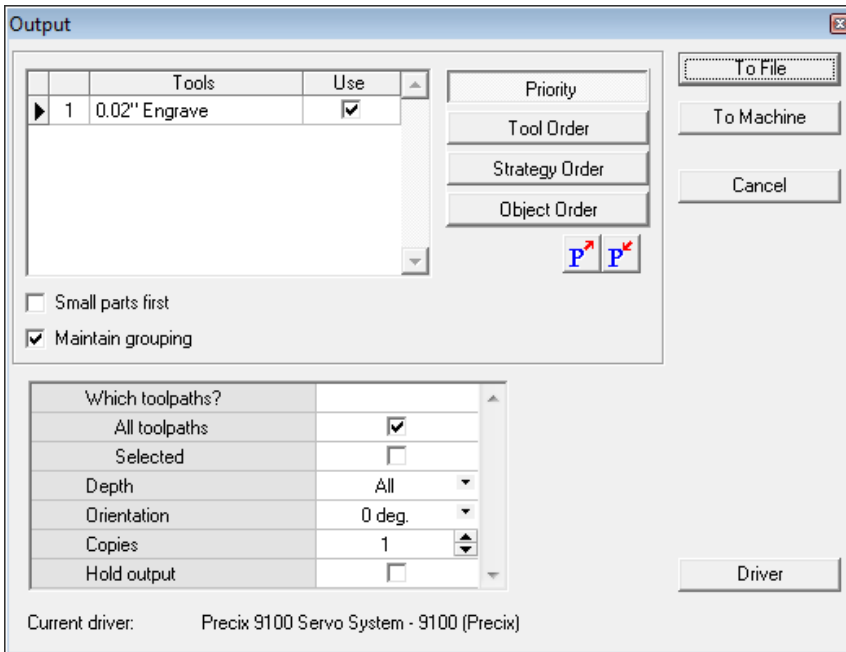
Generating Output

Menu: Machining / Output

Toolbar: Output / Output Options


The goal of virtually all the work you do in the software is creating toolpaths that will then be used to create an output file, or be sent directly to your router, engraver, or milling machine. After toolpaths have been created, the Output dialog determines which toolpaths will be sent, their order, and a number of other aspects of the output process.

Click **Output**. 



Generating output consists of the following steps:

1. Selecting the machine driver to use when outputting the design.
2. Determining the cutting order of the cuts that make up the design.
3. Selecting output parameters.
4. Outputting the design either to the Output Control Center or a file.


 Sending output can be as simple as opening the Output dialog and then clicking either **To File** or **To Machine**. The machine driver, cutting order options and output parameters are there to provide flexibility, not to make the process more complicated.

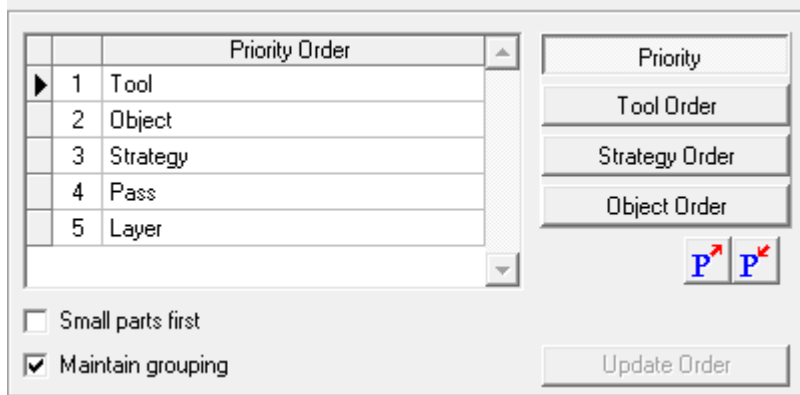
Selecting the Machine Driver

Click **Driver** to select the desired machine driver from the Driver Configuration dialog. For details on the Driver Configuration dialog, see “Configuring Machine Drivers”.

Determining the Cutting Order


The order in which the cuts are made determines the amount of time that it will take to output a finished piece. The software determines the cutting order based on a number of different parameters.


 It is not necessary to make any changes to the Ordering Options in order to successfully generate output. The default ordering options are sufficient to generate a high-quality finished piece. They only need to be changed if you want to fine-tune the process.



The ordering options can be set using the **Ordering Options** list and its related buttons. The buttons cause the list to display ordering options at four different levels:

- 🕒 **Priority** is the highest level, and allows you to set the priority levels of the major parameters relative to each other.
- 🕒 **Tool Order** sets the relative priority of the various types of tools used in the cutting passes.
- 🕒 **Strategy Order** sets the relative priority of the various strategies present in the design.
- 🕒 **Sort Method** sets which of the different automatic toolpath sorting methods is in use.
- 🕒 **Small Part First** – Check this box if you would like the small parts to cut out first.
- 🕒 **Maintain Grouping** – Check this box to treat group objects as one in reference to the cut order.

🕒 **Reset Parameters using Preferences**  - Click here to reset the Parameters using the Preferences.

🕒 **Save Current Parameters to Preferences**  – Click here to save the current parameters to the Preferences.

📄 You can also access the Ordering Options table from the 2D Simulation and Ortho Simulation tools. This allows you to make changes to the ordering options, and then immediately see how they will affect that final output. Changes you make to the ordering options in 2D Simulation and Ortho Simulation will be carried over to the Output dialog.

Setting the Priority

Clicking on the **Priority** button displays the **Priority Order** list, which lists the following parameters that can be used to sort the cuts in the design:

- Tool** The type of cutting too used to make the cut.
- Object** The *object* that the cut forms a part of. An object in this context means a separate piece that will be cut from the plate.
- Strategy** The strategy to which the cut belongs.

- Bundle** The *bundle* to which the cut belongs. Bundles are groups of toolpaths that make up a cutting pass.
- Pass** The cutting pass to which the cut belongs.

For example, if the priorities are in the order listed above, the cuts will be sorted first by the type of tool they use, then by the individual contours to which the cuts of each tool type belong.

Reordering the Priority List Entries

Entries in the list can be reordered by dragging them up or down in the list. Click on the row header on the left edge of the entry to select it, then drag the entry up or down by the header.

Setting the Priority to Minimize Tool Changes

The default Priority is designed to minimize tool changes in the output:

1. Tool
2. Object
3. Strategy
4. Bundle
5. Pass

If your output uses two tools, say a ¼" end mill and a 90-degree conic, all of the toolpaths associated with the first tool would come first, followed by all of the toolpaths associated with the second tool.

Setting the Priority to Cut Each Object Completely Before Continuing

There may be times when minimizing tool changes is not the most important thing to you. For example, the output may contain a number of toolpath groups and it is more important that you cut out each piece one at a time in order to allow you to progress with other work in the shop. In that case, you may want to move 'Object' to the top of the priority list. If each object (toolpath group) uses both the ¼" end mill and the 90-degree conic from the example above, there would be a tool change required as each object cuts out. This might increase the number of tool changes significantly, but it would allow you to get finished pieces off the table before the entire job is complete.

Returning to the Default Priority Settings

Click **Default Order** to return the Priority settings to their default order.

Setting the Tool Order

Clicking the **Tool Order** button displays the **Tools** list, which lists the priorities assigned to the different tool types used in the design. These settings are used to sort the cuts by tool type in the **Tool** stage of the sorting process.

For example, if a ¼" end mill tool will be used to cut separate pieces out of the plate, you should make that tool the lowest priority so that any other cuts that need to be made to the piece will be made before it is separated from the plate.

Reordering Tools

Drag the tool entries up and down in the list to assign their priorities. Click on the row header on the left edge of the entry to select it, then drag the entry up or down by the header.

Defining Tools as Unused

Clearing the check box in the **Use** column to the right of a tool will prevent any cuts that use that tool from being output.

Setting the Strategy Order

Clicking the **Strategy Order** button displays the **Strategies** list, which lists the priorities assigned to the different toolpath strategies that appear in the design. These settings are used to sort the cuts by strategy in the **Strategy** stage of the sorting process.

For example, if you used the same contours to create both male and female Routing Offset toolpaths for an inlay job, you would not want to send both strategies to the machine at the same time, because you would use different material to cut out the male offset than you would for the female offset. So, you would move one of the strategies lower on the list, or even mark it unused.

Drag the strategy entries up and down in the list to assign their priorities. Click on the row header on the left edge of the entry to select it, then drag the entry up or down by the header.

Defining Strategies as Unused

Clearing the check box in the **Use** column to the right of a strategy will prevent any cuts from that strategy from being output.

Setting the Toolpath Sort Method

Clicking the **Sort Method** button displays the **Sort Methods** list, which allows you to select a sorting method that will be used to automatically order the toolpath groups in the design. This is exactly the same as the ordering that can be done using the Order Toolpaths dialog.

If you have already ordered the toolpath groups manually, none of the methods will be checked.

If you check a method, it will automatically be applied to the toolpath groups in your design.

For details on the various sorting methods, see “Setting the Toolpath Order”.

Selecting Output Parameters

The following output parameters are available:

Which toolpaths	Select one of the following options:
All toolpaths	All toolpaths will be output.
Selected only	Only the currently selected toolpaths will be output.
Depth	Select one of the following options:
All	Toolpaths at all depths will be output.
[Specific Depth]	Only the cut passes at the specified depth will be output. This might be done to finish a job that had been interrupted near its end, for instance.
Orientation	Allows you to output the toolpaths rotated by some increment of 90 degrees. Select 0 , 90 , 180 , or 270 degrees.
Copies	This option is not implemented at this time. Leave it at 1 .

Hold Output

If you are sending the output to a machine through the Output Control Center, this option tells the OCC to hold the output in a queue so that you can send the queue of jobs to the machine when you are ready.

Outputting the Design

Outputting the Design to a File

To output the design to a file:

- 1. Click **To File**.
- 2. In the standard Windows Save As dialog that opens, specify the file name for the output file and click **OK**.

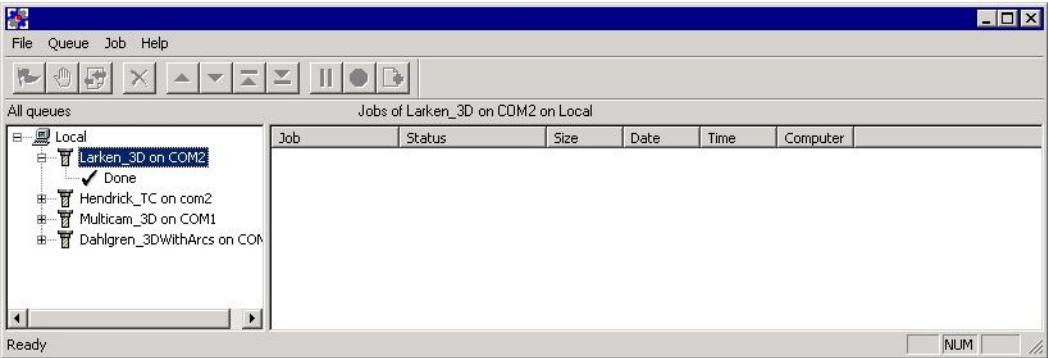
Outputting the Design to a Machine

To output the design to a machine, click **To Machine**.

The design will be created as a job in the Output Control Center queue for the selected machine.

Using Output Control Center to Manage Output

The Output Control Center (OCC) is used to communicate and manage your plotter, engraver and large format printer files.



The left side of the dialog displays the queues for all of the machines that have been set up. Clicking on the plus sign next to the queue expands it to show the **Done** queue for that machine.

When you select a queue from the left side, the right side will display any jobs that are queued to be sent to this machine. The following information is displayed for each job:

Job	The file name of the design that is being output.
Status	The current status of the job:
Ready	The job is ready to process
Hold	The job is being held (the Hold Job parameter was selected when it was output).

Processing	The job is being output on the machine. A percentage of the job that has been output is displayed.
Error	There was some error when communicating with the machine.
Size	The size of the job file.
Date	The date the job was sent on.
Time	The time the job was sent.
Computer	The computer the job was sent from.

Outputting Jobs

When you choose the Plot or RIP Print command, Inspire places a “job” in the queue folder of the target machine. The jobs are placed in the queue one after the other, in the order of their arrival.


The OCC automatically sends the jobs to the machine for output. The jobs are automatically sent in the order they are received.

Held Jobs

The Output dialog has a parameter called **Hold Job**. If you choose this parameter, the job will be created and placed in the queue, but it will not be processed and output until you release it using the **Release** command.


Holding Jobs

To hold a job that is currently in the output queue:

1. Select the job.
2. From the **Job** menu select **Hold**. 

Releasing Held Jobs


To release a held job:

1. Select the job.
2. From the **Job** menu select **Release**. 

Using the Done Queue to Repeat Output

Each machine's queue has a sub-queue called **Done**. When a job has been processed and output, it is moved to the **Done** queue, and the OCC starts processing the next job. Jobs in the Done queue can be sent to the machine to be output again. The main advantage of this is that the jobs do not need to be processed a second time, which can save time for large and complex jobs.


To output a job from the **Done** queue:

1. Select the job.
2. From the **Job** menu select **ReEngrave**. 

You can control the number of jobs kept in the Done queue through the Options command.


Aborting Processing of a Job

To abort processing of a job:

1. Select the job.
2. From the **Job** menu select **Abort**. 

Deleting Jobs

To delete a job from a queue:

1. Select the job.
2. From the **Job** menu select **Delete**. 
3. Click **OK** to confirm deletion.

12.3D Surfaces and 3D Toolpaths

Menu: Surface

Toolbar: 3D Surfaces

Introduction

EnRoute Pro introduces a number of new concepts for creating designs and the toolpaths to cut out these designs. All of the previous functions have been oriented around working with two-dimensional (2D) geometry, and then creating either 2D or engraving toolpaths based on the 2D geometry. The Pro functions provide the ability to create three-dimensional (3D) surfaces and then create 3D toolpaths associated with these surfaces. The surfaces you create are compatible with all of the other design elements, and the working environment is no different than the environment for the 2D design. This allows you to easily utilize whatever type of design and toolpath strategy is most appropriate for your application.



This chapter provides a brief introduction to the concept of 3D surfaces, and then subsequent chapters provide detailed information on creating surfaces, manipulating and combining surfaces, and then creating toolpaths to machine these surfaces.

3D Surface Applications

There are certainly countless ways in which 3D surfaces can be utilized in commercial applications. The EnRoute Pro tools are designed primarily as creative tools, making them most useful for applications that require creative designs. This includes woodworking and woodcarving, signs, jewelry, mold-making, and any other application that requires creative design tools.

While EnRoute's 3D tools can be used to create precise surfaces, their primary application is not considered to be in creating 3D engineered parts for mechanical applications.

Creating 3D Surfaces

EnRoute provides an array of tools that are used to create 3D surfaces, which are referred to as **Reliefs**. Reliefs are created and modified using both 2D contours and 3D mesh objects. 2D Contours are used to sweep and spin profiles and cross sections and to extrude shapes to create surfaces. 3D mesh objects can be created in EnRoute, and then applied to reliefs to create a finished surface. EnRoute also allows you to import a number of different file formats, so mesh objects that are created in other applications can be used too.

Creating Output for 3D Surfaces


3D toolpaths are created in much the same way as 2D toolpaths in EnRoute. Hatch Fills and Island Fills are the most common strategy types that are used with reliefs, but other strategies like routing offsets and engraving can also be effectively used to machine a surface. When these strategies are used with a relief, some additional parameters become necessary to define how the toolpaths should be created. Otherwise, the process is the same as the processes defined in Chapter 6. Creating 3D toolpaths is discussed in detail in Chapter 19.

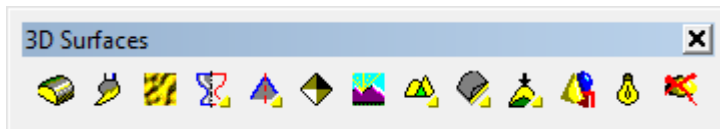
13.Creating Reliefs

Creating a Relief

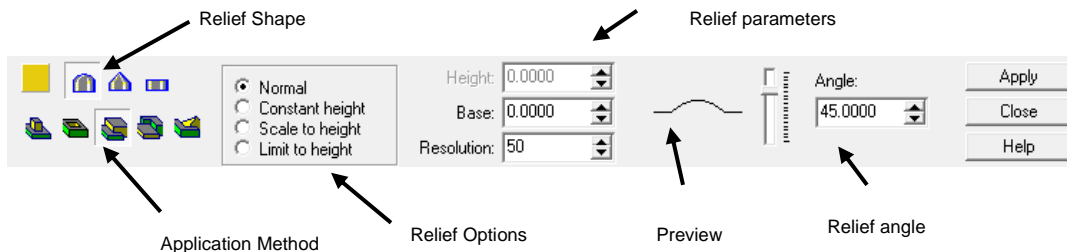
Menu: Surface / Create Relief

Toolbar: 3D Surfaces / Create Relief

The first step in working with surfaces is to create a relief, which can then be modified, and used to create 3D toolpaths. Click **Create Relief**. 



This will activate the **Precision Toolbar** for creating a relief. This dialog provides several parameters that allow you to create and modify reliefs in many different ways. The following sections provide a description of the parameters and how they affect the resulting relief.



Relief Shape

There are three basic shapes available to use for creating a relief. They provide the starting point for creating all of the different surfaces available in EnRoute.

Round



The round relief uses the relief angle to create a surface that is similar to a flexible surface that has been inflated. It can be used by itself to create rounded organic shapes, and it can also be effectively used to modify other surfaces.

Beveled



The beveled option creates a surface that is similar to the shape of a surface that is created by using Pyramid toolpaths. EnRoute creates this type of surface by extending a flat surface up from the perimeter of the selected contour to the 'center' of the shape where it meets a surface extended up from the other side of the shape. This is a complex operation, but EnRoute handles all of the computations, and all that is required is to provide the angle of the surface.

Flat



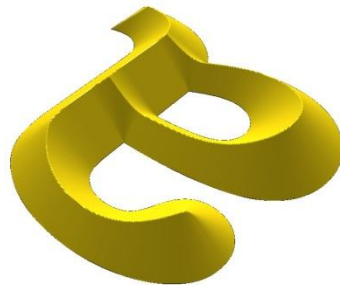
The flat option is used to create a simple flat surface. This option is used primarily as a means of creating a surface that can then be modified using other relief tools such as the sweeping and extrusion tools. By specifying a Height of 0.00, the flat surface is created at the height of the contours. If a height is specified, then the flat surface is created above that contours by that amount, as shown in the adjacent image.

Relief Options

The relief options allow you to modify the basic relief shapes to create different surface shapes. So, for a given relief shape, such as Round, you have the following four different options that allow you to modify how that shape is created.

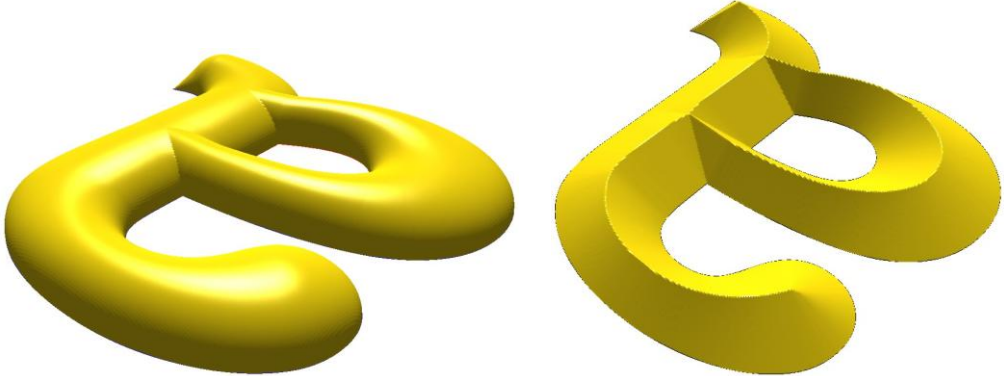
Normal

This is the default option that is described in the previous Relief Shape section. The key distinguishing characteristic of this option is that the relief shape, either round or beveled, is created based on the specified angle. The height of the relief is defined based on the width of the surface at each location. As the surface becomes wider or narrower, its height is greater or smaller accordingly. The following images illustrate this option for both round and beveled relief shapes.



Constant Height

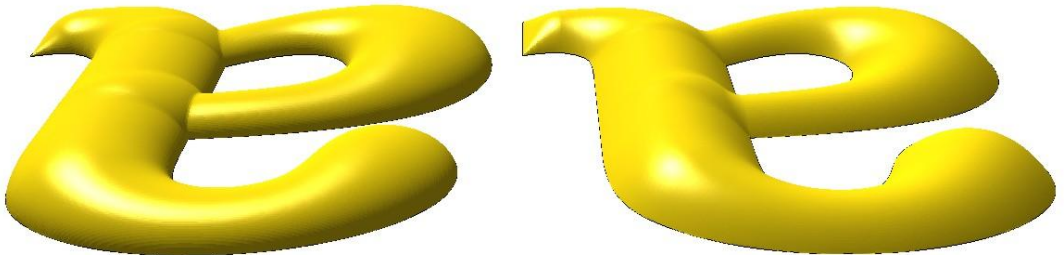
The Constant Height option changes the way that round and beveled shapes are created. Instead of using a relief angle to define the relative steepness of the surface, you define a height for the relief, and the surface is created to match that height. This option creates a surface that can be described as more mechanical looking. The way that it varies from the Normal option is most apparent in areas where different parts of a relief merge together. The following two graphics illustrate this option.



Scale to Height

This option is a variation of the Normal option. Using this option first creates the surface using the Normal option, and then the relief is scaled vertically so that its height matches the specified height. It comes in handy when you want to create normal relief, but you also want it to be a specific height. It is also a way to create surfaces that are subtly different by specifying different angles and the same height.

The following images show a relief that was created using the Scale to Height option. The relief on the left used a relief angle of 85 degrees and the relief on the right used an angle of 15 degrees. The height of each relief is the same, but the shape of each relief is different because of the different angle.



Limit to Height

This option is also a variation of the Normal option. The relief is first created using the normal option, and then it is cut off at the specified height. It is useful in cases where you want to create a relief that has rounded or beveled profile up to a certain height, and then a flat top surface.

The following graphic shows a relief that uses a rounded shape that is limited to a height that is about half the height that it would have been if it wasn't limited. This is an option that will not likely be used very often, but it can be useful for specific applications.



Application Method

The application method defines how the application of a relief modifies the surface. The terms for each option are descriptive of how the relief will be modified.

- The Application Method is a concept that is common throughout all of the relief creation tools. The way that the concept is used is consistent for each of these tools.

Add

This is the option that will likely be used most often. It is used to create a raised relief. If you are modifying an existing relief, the surface will be raised using the selected contours and parameters specified for the relief.

The following image shows an example of a relief created using the Add option.

In this image, the top surface was created using the Add option on an elliptical contour.

The bottom surface was created by adding the letters to an existing relief surface. This was accomplished by selecting the relief and the contours for the letters and then applying the relief function.



Subtract

This option is used to create a recessed relief. It can either be applied to contours to create new recessed reliefs, or using contours to modify an existing relief. If an existing relief is to be modified, it should be selected along with whatever contours are to be used to modify it.

The following graphic shows the effect of the subtract option.

The top portion of the image shows individual contours that were used to create individual reliefs. The bottom elliptical relief was modified using the same contours and the subtract option.



Merge Highest

This option is most effective when it is used to modify an existing relief. It can be used when creating a new relief, but the effect will be the same as the Add option. When it is used to modify a relief, the new portion of the relief only modifies the existing relief where it is higher than the existing relief.

This is best illustrated with a simple example, as shown in the following image.

In this example, the top letter was created using the Merge Highest option. Notice that it is the same as if it was created using the Add option.

The bottom portion of the image shows an existing elliptical relief that was modified using the same contour. The key thing to notice is that the relief was only modified where the relief created by the new contour was taller than the existing relief.



Note that if you are creating a new relief, the **Add**, **Merge Highest** and **Replace** options will all create the same result.




Merge Lowest

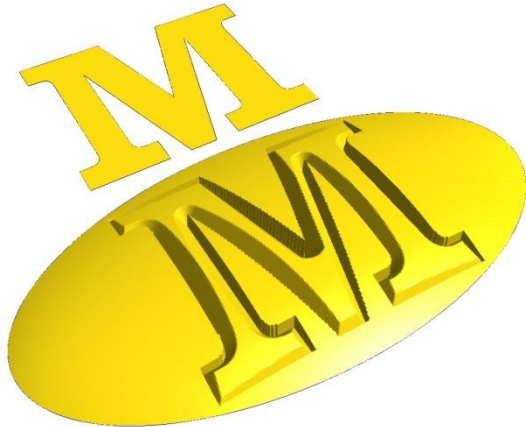
The Merge Lowest option is similar to Merge Highest, except that a relief is only modified where the new relief is lower than the existing relief. This option is a little less intuitive than the Merge Highest option. Essentially, wherever a relief would not be modified by a Merge Highest option, it would be modified by a Merge Lowest option.

The following graphic demonstrates how a contour can be used to modify an existing relief using this option.

The top portion of the image shows the result of using this option to create a new relief. Note that the result is a flat relief. This result is caused by the fact that a flat surface is the lowest surface over the entire area of the relief, so that is the result. This indicates that this option should not be used when creating new reliefs.

The bottom image demonstrates the result of modifying an existing relief using the same contour. The relief is modified only in areas where the new profile is lower than the existing relief. In areas where the new profile is higher than the existing relief, the existing relief is not modified. This option will likely be appropriate only in special applications.

 Note that using the **Merge Lowest** option to create a new relief will result in a flat surface being created.



Replace

The Replace option is also used primarily to modify an existing relief. If it is used to create a new relief, the result is the same as the Add option. When it is used to modify a relief, the applied profile takes the place of the existing relief wherever the new profile overlaps the existing profile.

The following image shows the result of the Replace option used to create and modify a relief.

In this image, the top letter was created as a new relief. The bottom relief was modified using the contour of the letter. You can see that the relief created by the letter takes the place, or replaces, the existing relief.



Relief Parameters

Height

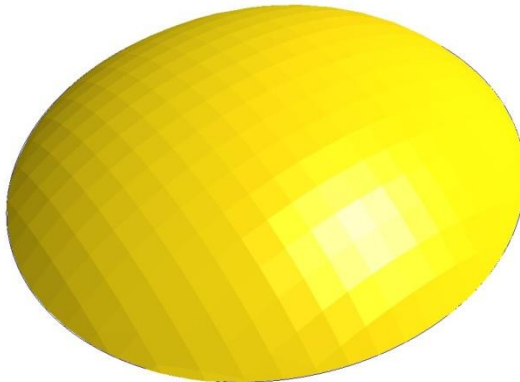
The Height parameter is used to define the height of the relief that is created when appropriate for the selected relief option. A height is used for the Constant Height, Scale to Height, and Limit to Height options. The entered value uses the active units.

EnRoute enables and disables this parameter based on the Relief Option selected, so when it is not necessary for a value to be entered, this parameter is disabled. When the Normal option is selected, the relief is defined using the angle parameter, so the height is not needed.

Resolution

This parameter is used by EnRoute when a new relief is being created with this command. The Resolution is specified in 'dots per inch' (dpi) regardless of the units that are selected. So, even if you are operating EnRoute with active unit of millimeters, this parameter is still specified in dpi.

The resolution of the relief defines the size of the grid units that make up the relief. A relief is a gridded surface and when a relief is created, it is necessary to define how large each grid unit should be. If a resolution of 100 dpi is selected, then each grid unit will be 1/100 inches wide and tall. Therefore, the level of detail for the information on that relief will be limited 0.01 inch. This resolution is typically adequate for most surfaces in EnRoute, unless you are creating a small object that requires more detail.



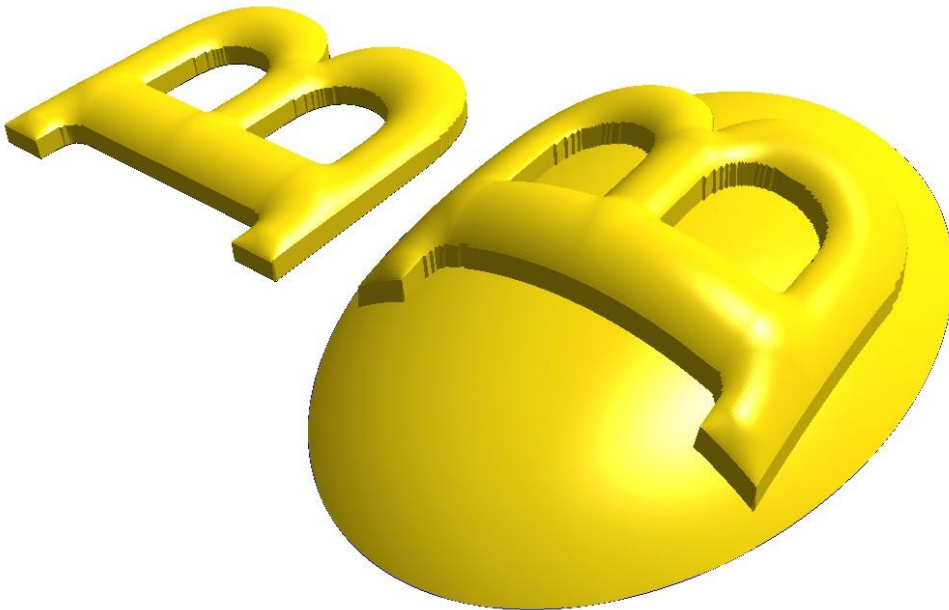
The thing to remember is that reliefs in your drawing require memory that is directly related to the size of the relief and its resolution. As you increase the size of a relief, it requires more memory. If two reliefs are the same size and one relief is 100 dpi and the other 200 dpi, the second relief will require four times the amount of memory as the first. You can see that this is an important consideration when defining a relief. In general, you should define as low a resolution for a relief as will accommodate the level of detail for that relief.

Base

The Base parameter allows you to move the relief surface up, creating vertical sides on the portion of the relief that you are modifying. If you are creating a new relief, a base dimension will create vertical sides around the perimeter of the relief.

This parameter will likely be most useful when you are modifying an existing relief and would like to include vertical sides around the portion you are modifying. If you are creating a new relief and want it to have vertical sides, it is possible to move it vertically to position it in the plate so that when it is cut out the parts will have the correct height of vertical edges.

The following image shows two reliefs that include base dimensions. The top relief was created with a rounded profile and a base dimension to create the vertical sides on the letter. The bottom image was a relief that was modified to add the letter with the base dimension.



Angle

The Angle parameter is used to define Round and Beveled reliefs when the Normal relief option is selected. Both positive and negative angles may be used between 0 and 90. With beveled reliefs, the angle defines the angle of the bevel. With round reliefs, the angle defines the angle of a line tangent to the circle that defines relief. If an angle of 90 degrees is used with a round relief, a semi-circle is used to create the relief.

The preview drawing in the dialog shows a small preview of the relief shape based on the angle that is defined. The angle can be entered directly into the edit box, or the spinner arrows can be used to define the angle, and there is also a slider bar that can be used to define the angle.

The following graphic show how different effects can easily be achieved just by using different angles. Relatively flat surfaces such as those on the left are created by using small angles, and more dramatic surfaces are created by using larger angles such as those on the right.

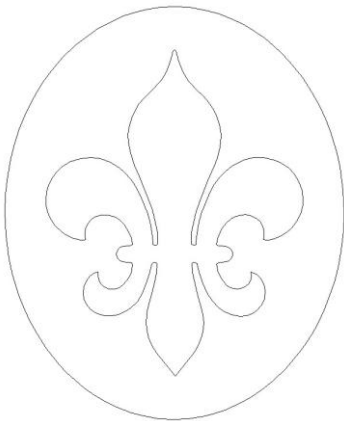


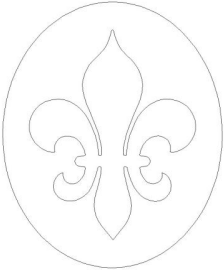
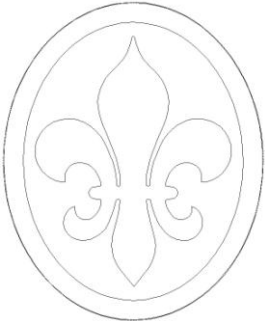
📄 When the Constant Height relief option is selected, the Angle parameter is disabled because it is not used in this option.

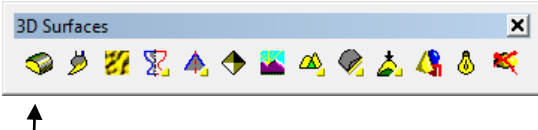


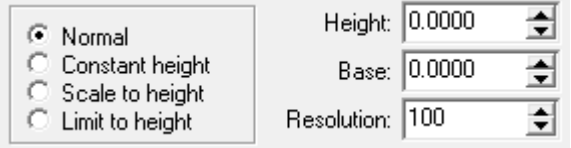

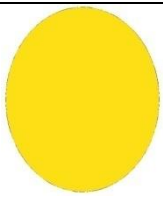



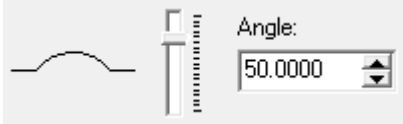
Using the Relief Dialog





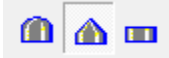

Now that the options and parameters have been introduced in the previous sections, the next task is to use the Relief creation tool to create and modify a relief. This section includes a presentation of the steps required to create a relief using just this tool.


The following graphic shows some simple 2D artwork that is used to create the relief example. To repeat this example, you can create an ellipse and some contour artwork similar to the 'fleur' shown in this example.



<div>1. Create your 2D artwork. In this example, create an ellipse and center artwork like the fleur. The ellipse is approximately 8 inches tall by 6.5 inches wide (200mm x 116mm).</div>	
<div>2. Select the ellipse and offset it with a 0.5 inch inline and a 0.01 inch outline.</div>	

3. Select the original ellipse. This will be used to create the base relief.	
4. Click on the Create Relief icon, or select the Surface menu and Create Relief.	
5. Select the Flat relief option.	
6. Select the Add option.	
7. Define the parameters to Height = 0.00, Base = 0.00 and Resolution = 100 and select the Normal relief type.	
8. Click Apply to get the following result. Click the Rendered Icon  to toggle to the rendered view of the relief on and off.	
9. Now select the relief and the two offset ellipses that were created in Step 2. (Toggle the rendered view  off in order to select the ellipses).	
10. Select the Round relief option.	
11. Select the Add Option and leave the relief type on Normal.	
12. Set the Angle to 50 degrees.	

13. Click Apply to create the next step in the finished relief.	
14. Now select the relief and the smaller ellipse.	
15. Select the Round relief option.	
16. Select the Subtract application method.	
17. Set the Angle to 30 degrees.	
18. Click Apply to create the next portion of the relief.	
19. Now select the relief and the center artwork.	
20. Select the Beveled option	
21. Select the Add application method.	

<p>22. Define the parameters to Height = 0.30, Base = 0.00 and select the Constant Height relief type.</p>	<div data-bbox="646 115 899 258"> <input type="radio"/> Normal <input checked="" type="radio"/> Constant height <input type="radio"/> Scale to height <input type="radio"/> Limit to height </div> <div data-bbox="926 115 1206 258"> Height: 0.3000 Base: 0.0000 Resolution: 100 </div>
<p>23. Click Apply to create the finished relief.</p>	

As you can see, using just the **Create Relief** tool provides some very powerful capabilities for creating finished reliefs in EnRoute. The following chapters discuss the many other tools EnRoute provides for creating and modifying surfaces. The concepts presented in this chapter regarding adding and subtracting reliefs are consistent throughout each of the other functions.



14.Revolutions and Extrusions

Introduction

The previous chapter provides the basic tools for creating relief surfaces. This chapter describes additional tools that increase your ability to define intricate surfaces. These tools use revolutions and extrusions to create meshed objects and modify selected relief surfaces.

When modifying reliefs, the concepts presented in the previous chapter regarding application methods work identically with these tools. The primary new concept with these tools is that EnRoute uses wizards to prompt you regarding the selection of appropriate contours to use to define how the function is to be applied. After you understand how they work, they provide a flexible method for completing each function.

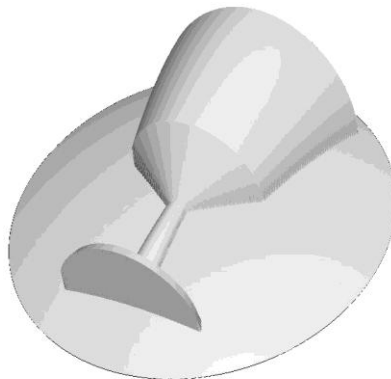
Revolve

Menu: Surface / Create Surfaces / Revolve

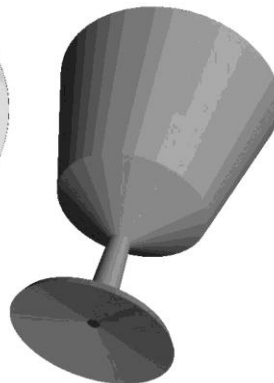
Toolbar: 3D Surfaces / Revolve



The Revolve tool allows you to use one or more contours to revolve around an axis to either modify a relief or to create a new mesh object, as illustrated in the following image.




Relief Surface

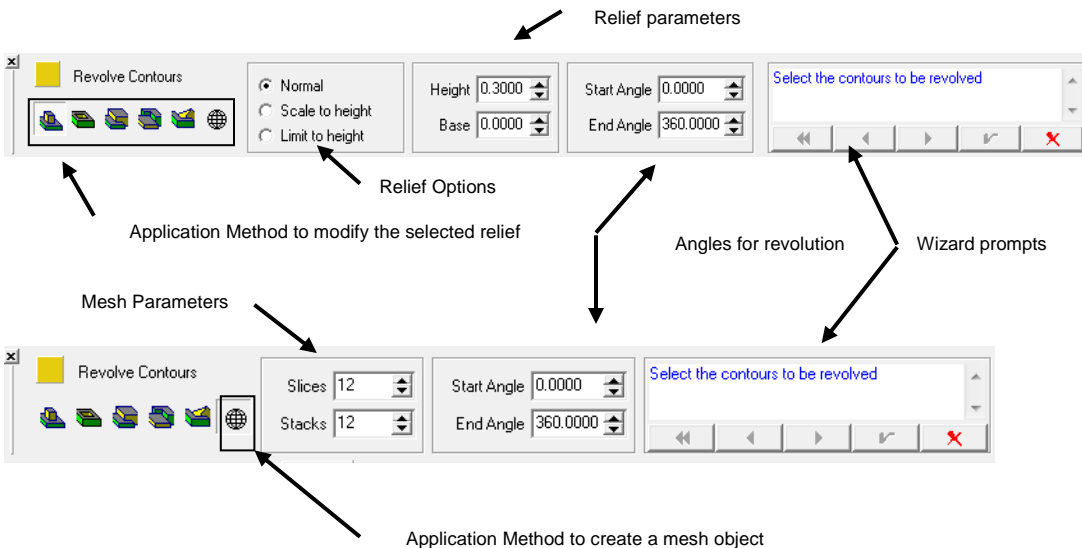


Mesh Object

Revolve Dialog

Selecting the **Revolve** icon , or selecting the **Surface** menu and then **Create Surface** and **Revolve**, will activate the Revolve dialog. This dialog allows you to select the appropriate options and parameters required to create the new surface.

The revolve dialog provides two sets of parameters depending on whether you have selected the option of creating a mesh object, or any other application method for modifying an existing relief. If you select an application method for modifying a relief, the following dialog is presented.




Application Method

All of the options for applying the results of the revolve operation are the same as the options described in the previous chapter on relief creation. The options for Add, Subtract, Merge Highest,



Merge Lowest, and Replace all apply to the situation where the revolution is to be applied to an existing relief. For this to be possible, it is necessary to select the relief to be modified before starting the Revolve command.

The right-most application method option  allows you to create a mesh object with the Revolve command. As shown in the dialogs above, when this option is selected, the parameters required for the command are different than when a relief is to be modified. When this option is selected, the parameters change, providing space to enter the number of slices and stacks that will make up the new object. These are described below, and the rest of the revolve command is the same whether you are modifying a relief or creating a new mesh object.

Relief Options

These options function similar to how they work in the relief creation tool. In this case you have three options, Normal, Scale to Height, and Limit to Height.

- **Normal** – The revolved surface is created just as the selected profile and axis define it, without any vertical scaling.
- **Scale to Height** – The revolved surface is scaled to match the height parameter as it is applied to the relief. This allows you to create a revolved surface and scale it in one step.
- **Limit to Height** – The revolved surface is limited in height by the height parameter. If a portion of the revolved surface extends above this height, it is truncated before it is applied to the selected relief.

Relief Parameters

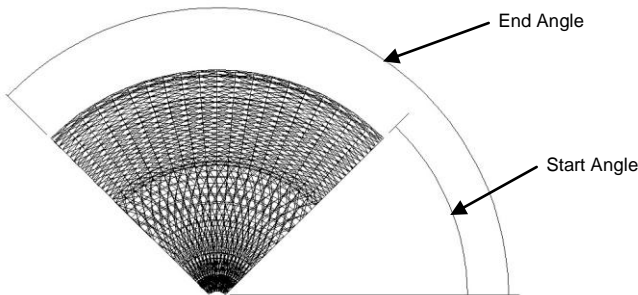
The relief parameters of **Height** and **Base** are available when modifying a relief.

- **Height** – This value defines the height if either the 'Scale to Height' or 'Limit to Height' relief option is selected.
- **Base** – This value allows you to define a base height that can be added to the revolved surface as it is applied to the selected relief.

Angles for Revolution

The **Start Angle** and **Finish Angle** define the portion of a complete 360-degree revolution that will be completed. A complete revolution would have a start angle of 0 degrees and an end angle of 360 degrees. A half revolution would use a start angle of 0 degrees and an end angle of 180 degrees.

The following front view of a revolved mesh object illustrates the start and finish angles. This object used a start angle of 45 degrees and an end angle of 135 degrees.








Wizard Prompts

The **Revolve** tool requires more than one step in order to complete it. This portion of the dialog contains an area that contains a prompt that tells you the next step required in order to complete the function. It also contains buttons that allow you to move from step to step, back up a step, back up to that start of the function, execute the function, or exit the function.

The first prompt asks you to ‘**Select the contours to be revolved.**’ After selecting one or more contours, you can select the next step button, which will then prompt you to ‘**Select the axis of revolution (Optional)**’. At this stage, you can either select a contour that will serve as the axis of revolution, or select the execute button.

Following is a listing of each of the wizard buttons and its function.

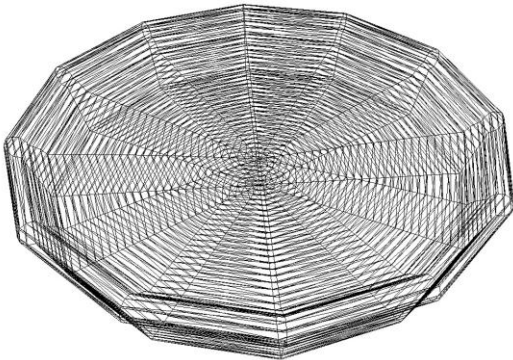
	Start	Return to the start of the command
	Back	Go back one step
	Next	Go to the next step
	Execute	Execute the function
	Cancel	Cancel the function

Mesh Parameters

When the mesh option is selected, it activates parameters that are specific to this option. They include the number of **Slices** and the number of **Stacks** that are to be used to create the mesh object.

A mesh object is composed of triangles, or facets, that make up its surface. It is necessary to instruct EnRoute regarding how many of these facets should be used to construct the object. The number of **Stacks** is the number of sections that are used along the length of the profile that is revolved. The **Slices** value is the number of sections that are used between the start and end angles.

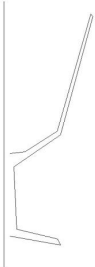
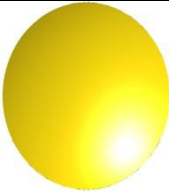





The following graphic illustrates this concept. This object was created to resemble a pie, with each of the pieces of the pie representing a slice, and the facets that make up each piece of the pie representing the stacks. The profile of a piece of the pie was revolved around the center of the pie.


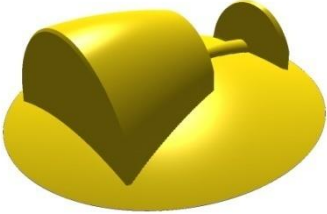


Modify a Relief Surface

The first option is to use the Revolve tool to modify an existing relief surface. The advantage of this option is that it takes care of creating the revolution and applying it to the relief in one step. Also, it automatically takes care of creating a smooth surface without you having to specify any additional parameters. The limitation of this option is that it does not allow you to modify the revolved surface before applying it to the relief.

The following steps lead you through the process of creating a revolution to modify an existing relief.

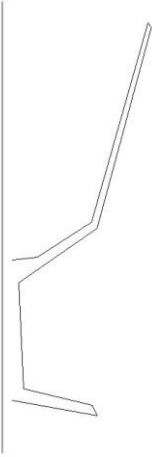


1. Create your 2D artwork. In this example, create an ellipse, a vertical line, and a profile like the one shown. The ellipse is approximately 8.5 inches tall by 7.75 inches wide (215mm x 195mm).	
2. Select the ellipse and use it to create a round relief using an angle of 25 degrees.	
3. Select the relief, and then select the Revolve icon  , or select the Surface menu and then Create Surface/Revolve.	
4. Click on the Add icon to select that option	
5. Select the Normal application method	
6. Define a Start Angle of 0.0 and an End Angle of 180.00	
7. Click to select the profile contour you want to revolve, and then click the Next Step icon  in the wizard.	
8. Click on the vertical contour to identify the axis of revolution. This step is optional.	



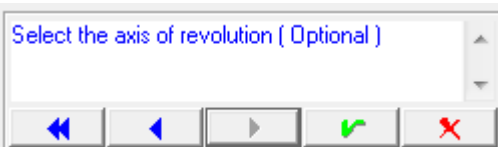


9. Click on the Execute button to complete the command.	
10. The following shows a rendered perspective view of the resulting relief.	

Create a Mesh Surface

Creating a mesh surface using the Revolve tool is almost the same as using this tool to modify a relief. Key differences are that it is not necessary to identify a relief to modify, you need to select the mesh object option, and then you need to define the mesh parameters of slices and stacks. The resulting mesh object can then be used to modify a relief, but first you can scale, rotate and position the object so that when you add it to a relief it has the effect you need. You can also define the number of slices and stacks, using the faceted nature of the object to your advantage.

The following steps illustrate the method of creating a mesh object using the Revolve tool.

1. Create your 2D artwork. In this example, create a vertical line and a profile like the one shown.	
2. Select the Revolve icon  , or select the Surface menu and then Create Surface/Revolve.	
3. Click on the Mesh icon to select that option.	

4. Define Slices = 32 and Stacks = 24	
5. Define a Start Angle of 0.0 and an End Angle of 180.00	
6. Click to select the profile contour you want to revolve, and then click the Next Step icon  in the wizard.	
7. Click on the vertical contour to identify the axis of revolution. (This step is optional.)	
8. Click on the Execute button to complete the command.	
9. The following shows the mesh object like the one that will be created. This object can then be scaled and rotated, and then used to modify a relief.	

Spin

Menu: Surface / Create Surfaces / Spin

Toolbar: 3D Surfaces / Revolve / Spin




The **Spin Tool** is quite similar to the **Revolve Tool** that was explained in the previous section. The primary difference is that in the **Spin tool**, the selected contours are spun around the Z-axis, located at a center of rotation by the user. In addition, there are a couple of options that are introduced in the spin tool that give you some additional control over the shape of the resulting surface.

You have the option to apply a spin directly to a relief, or to create a mesh surface. As with this option in other tools, when the spin is applied to a relief it is automatically smoothed and when you create a mesh surface, the result is an object that can be modified further before applying it to a relief.

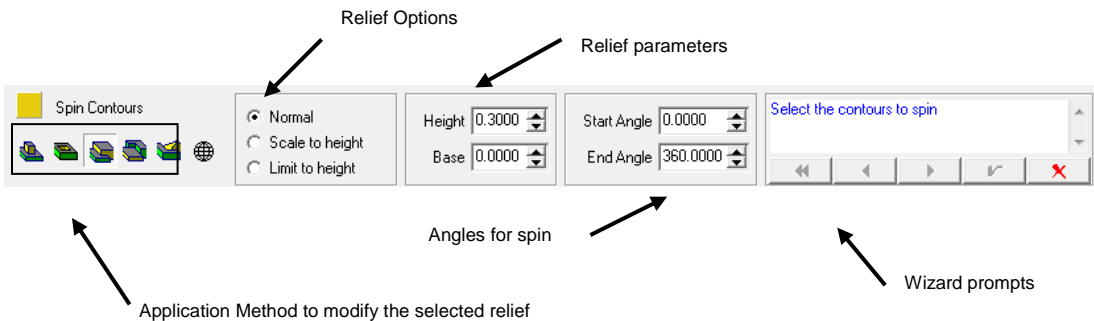
With some careful planning, the spin tool can be used to create intricate and interesting surfaces.



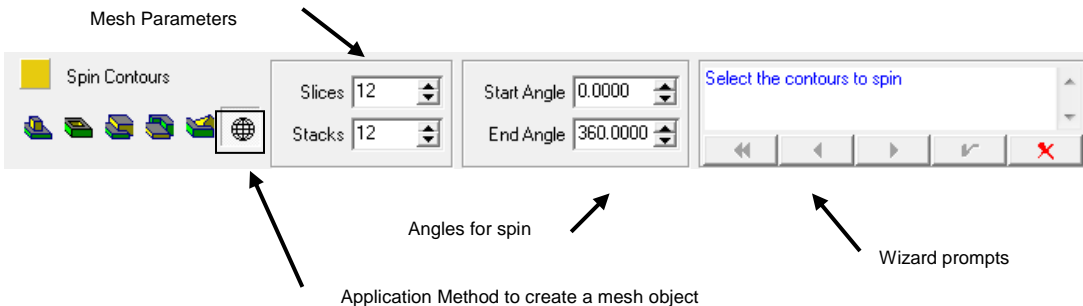
Spin Dialog

Selecting the **Spin** icon , or selecting the **Surface** menu and then **Create Surface** and **Spin**, will activate the Spin dialog. This dialog allows you to select the appropriate options and parameters required to create the new surface.

The Spin dialog provides two sets of parameters depending on whether you have selected the option of creating a mesh object, or any other application method for modifying an existing relief. If you select an application method for modifying a relief, the following dialog is presented.



If you select the **Application Method** for creating a mesh object, the dialog is changed to allow you to enter the proper parameters for the mesh, and shown below.




Application Method

All of the options for applying the results of the spin operation are the same as the options described in the chapter on relief creation. The options for Add, Subtract, Merge Highest, Merge Lowest, and



Replace all apply to the situation where the revolution is to be applied to an existing relief. For this to be possible, it is necessary to select the relief to be modified before starting the Spin command.

The right-most application method option  allows you to create a mesh object with the Spin command. As shown in the dialogs above, when this option is selected, the parameters required for the command are different than when a relief is to be modified. When this option is selected, the parameters change, providing space to enter the number of slices and stacks that will make up the new object. These are described below, and the rest of the Spin command is the same whether you are modifying a relief or creating a new mesh object.

Relief Options

These options function similar to how they work in the relief creation tool. In this case you have three options, Normal, Scale to Height, and Limit to Height.

- **Normal** – The new spin surface is created just as the selected profile and other options define it, without any vertical scaling.
- **Scale to Height** – The new spin surface is scaled to match the height parameter as it is applied to the relief. This allows you to create a surface and scale it in one step.
- **Limit to Height** – The new spin surface is limited in height by the height parameter. If a portion of the new surface extends above this height, it is truncated before it is applied to the selected relief.

Relief Parameters

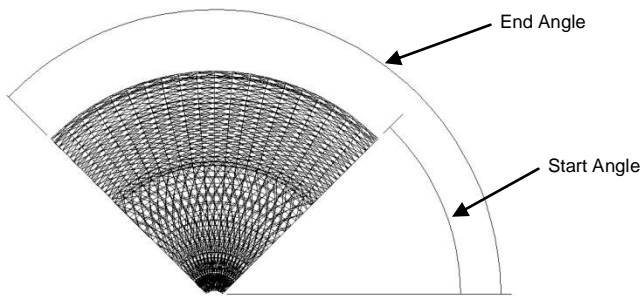
The relief parameters of **Height** and **Base** are available when modifying a relief.

- **Height** – This value defines the height if either the ‘Scale to Height’ or ‘Limit to Height’ relief option is selected.
- **Base** – This value allows you to define a base height that can be added to the revolved surface as it is applied to the selected relief.

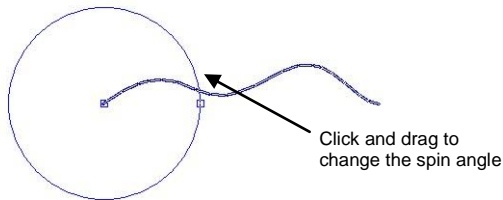
Spin Angles

The **Start Angle** and **Finish Angle** define the portion of a complete 360-degree spin surface that will be completed. A complete revolution would have a start angle of 0 degrees and an end angle of 360 degrees. A half revolution would use a start angle of 0 degrees and an end angle of 180 degrees.


The following top view of a mesh object illustrates the start and finish angles. This object used a start angle of 45 degrees and an end angle of 135 degrees.



In addition to defining the start and end angles using the dialog, the Spin tool also allows you to interactively define this angle by clicking and dragging on the circle that is used to define the center of the spin. This circle is shown in the image below.

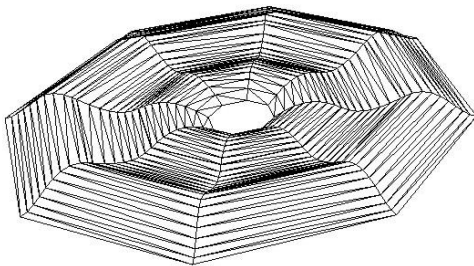


Mesh Parameters

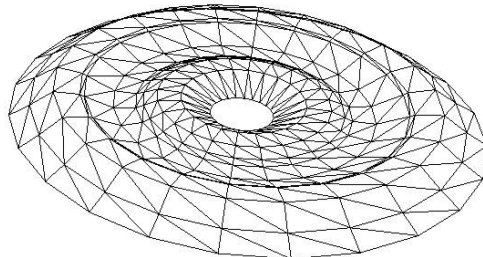
When the mesh option  is selected, it activates parameters that are specific to this option. They include the number of **Slices** and the number of **Stacks** that are to be used to create the mesh object.

A mesh object is composed of triangles, or facets, that make up its surface. It is necessary to instruct EnRoute regarding how many of these facets should be used to construct the object. The number of **Slices** is the number of sections that are used along the length of the profile that is spun. The **Stacks** value is the number of sections that are used between the start and end angles.

The following graphic illustrates this concept. The same profile was used to create these objects. The object on the left was created using more slices and fewer stacks, and the object on the right was created using fewer slices and more stacks.



Slices = 24
Stacks = 8



Slices = 8
Stacks = 24

Wizard Prompts






The Spin tool requires more than one step in order to complete it. This portion of the dialog contains an area that contains a prompt that tells you the next step required in order to complete the function. It also contains buttons that allow you to move from step to step, back up a step, back up to that start of the function, execute the function, or exit the function.



The following table lists the prompts in the Spin tool, along with an explanation the appropriate action.

Select the contours to spin	This is the first step, and you are prompted to select the contours you want to use as the profiles for the spin. Note that you can select more than one profile contour to use.
Select the height control curve (Optional)	This optional step allows you to select another curve to use to modify the height of the spin surface.
Select the width control curve (Optional)	This optional step allows you to select another curve to use to modify the width of the surface as it is spun around the center axis.

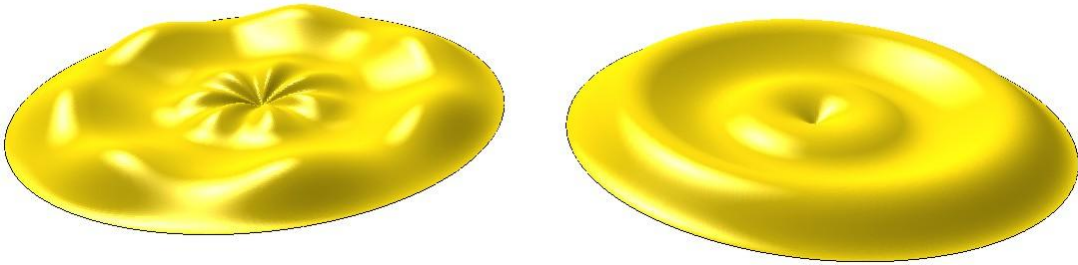
Following is a listing of each of the wizard buttons and its function.

	Start	Return to the start of the command
	Back	Go back one step
	Next	Go to the next step
	Execute	Execute the function
	Cancel	Cancel the function

Height Control Curve

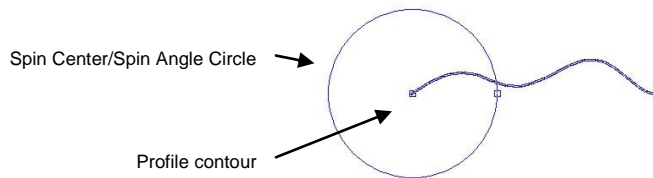
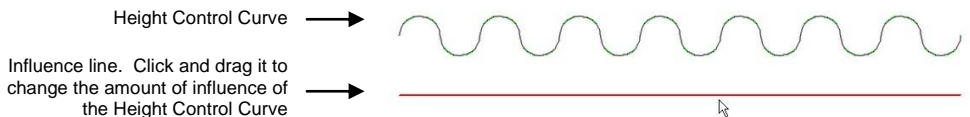
The optional second step of the wizard gives you the option to identify a curve that will modify the shape of the spun surface as it is created. This provides a creative tool that allows you to create a surface that can be quite different than the one you would get without this option.

The following graphic shows two surfaces created using this tool. The surface on the left was created using a **Height Control Curve**. The surface on the right used the same profile without using this step.



The next graphic shows the contours that were used to create the surfaces above. The **Height Control Curve** is a contour in EnRoute that is used to define the shape of how the surface is to be modified.

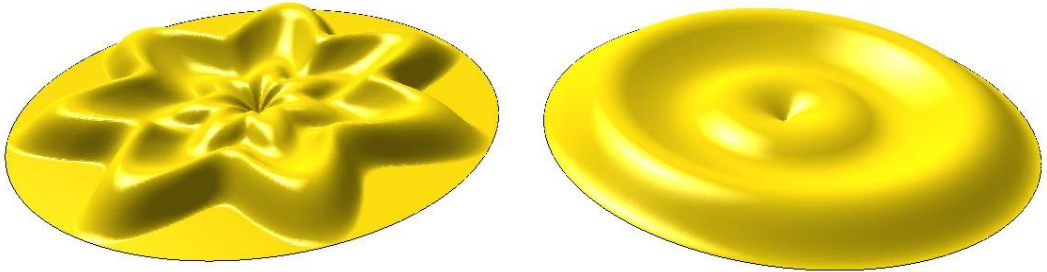
After selecting the **Height Control Curve**, the **Influence Line** is automatically drawn along the bottom of the curve. This line determines the amount of influence the **Height Control Curve** will have on the spun surface. As you interactively drag the **Influence Line** away from the **Height Control Curve**, the curve will have less influence on the spun surface as it is created.



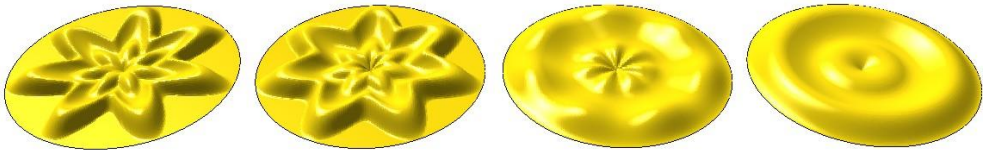
Width Control Curve

The optional third step of the wizard works similarly to the second step, except now you have the option of choosing a curve that will affect the width of the spun surface.

The following graphic shows two spun surfaces. The surface on the left was created using a **Width Control Curve**, and the surface on the right did not use this option.



The optional steps to modify the surface can be used separately, or together to create surfaces. The following image shows surfaces that were created using the same method, except for the use of these optional steps.



With both
options

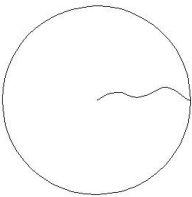
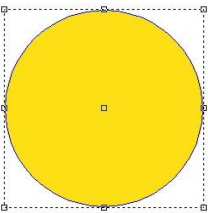


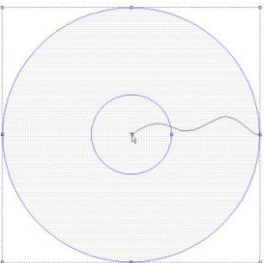

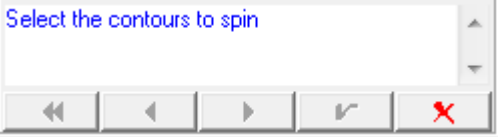
With Width
Control Curve

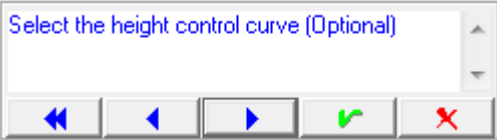

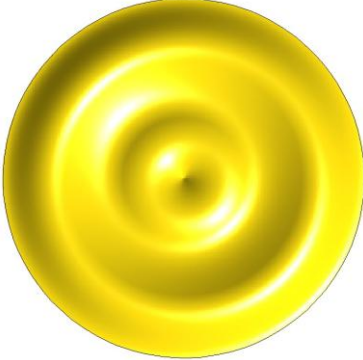
With Height
Control curve

Just the profile

Modify a Relief Surface

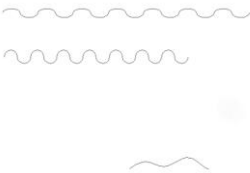



This section provides a table with the steps necessary to create a spin surface using the option of modifying an existing relief.

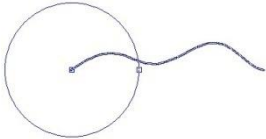

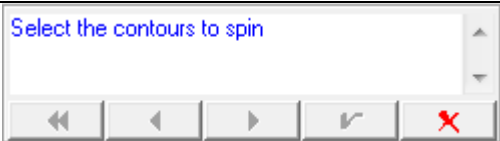
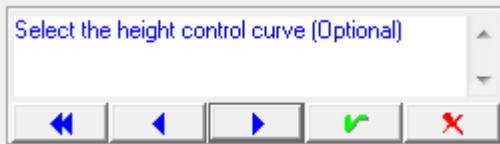
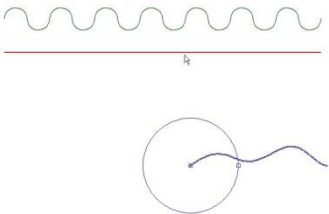

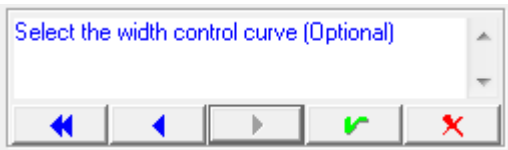
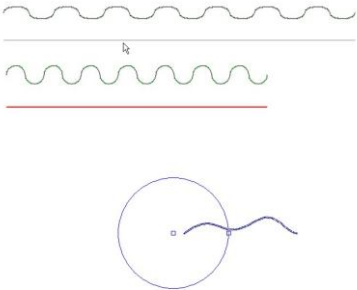
1. Create the 2D artwork. In this example, create a circle and a contour you want to use as the profile for the spin.	
2. Select the circle and use it to create a flat relief.	
3. Select the relief, and then select the Spin icon, or select the Surface menu and then Create Surface/Spin .	
4. Click on the Add icon to select that option.	
5. Select the Normal application method.	
6. Define a Start Angle of 0.0 and an End Angle of 360.00.	
7. Click and drag the blue centering circle to define the center point for the spin.	
8. Click to select the profile contour you want to spin, and then click the Next Step icon  in the wizard.	


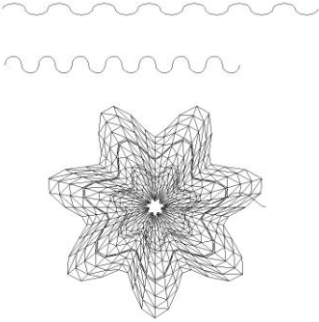
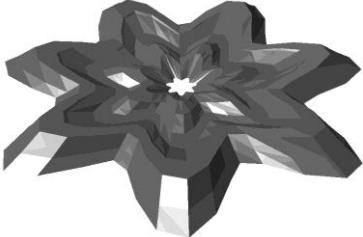
<p>9. If you want to include either of the optional steps, you can identify a Height Control Curve and/or a Width Control Curve.</p>	
<p>10. Click on the Execute button to complete the command.</p>	
<p>11. The following shows a rendered top view of the resulting relief.</p>	

Create a Mesh Surface

This next set of steps shows how to create a mesh object using the **Spin** command. It also demonstrates the two optional steps for selecting a **Height Control Curve** and a **Width Control Curve**.

<p>1. Create your 2D artwork. In this example, create the contour you want to use as the profile for the spin, and also create two additional curves to use as the Height Control Curve and the Width Control Curve.</p>	
<p>2. Select the relief, and then select the Spin  icon, or select the Surface menu and then Create Surface/Spin.</p>	
<p>3. Click on the Mesh icon to select that option.</p>	
<p>4. Define Slices = 24 and Stacks = 72.</p>	

5. Define a Start Angle of 0.0 and an End Angle of 360.00.	
6. Click and drag the blue centering circle to define the center point for the spin.	
7. Click to select the profile contour you want to revolve, and then click the Next Step icon  in the wizard.	
8. At this step, you are prompted to select the Height Control Curve.	
9. Click on the Height Control Curve . A red influence line is drawn; click and drag this line down to reduce the influence of the Height Control Curve.	
10. Click on the Next Step icon  to move to the next step in the wizard. You will be prompted to select a width control curve.	
11. Click on the Width Control Curve . A gray influence line is drawn; click and drag this line down to reduce the influence of the Width Control Curve.	

<p>12. Click on the Execute button to complete the command.</p>	
<p>13. An object similar to the one shown to the right will be created.</p>	
<p>14. This image shows a rendered view of the object. You can adjust the number of Slices and Stacks to change the 'smoothness' of the object.</p>	

Extrude

Menu: Surface / Create Surfaces / Extrude

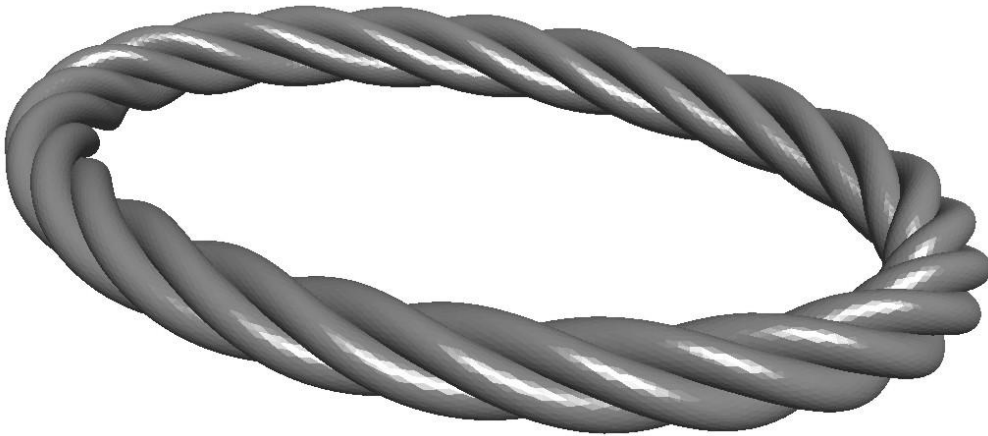
Toolbar: 3D Surfaces / Extrude




The Extrude tool provides a means to extrude a profile, or profiles, along a path to either modify a relief or create a mesh object. While the previous two tools are used to spin and revolve profiles around a central axis, this tool creates its surface along a path. Otherwise, it is very similar to the Spin and Revolve tools.

Two options that provide some additional flexibility include the ability to change the size of the profiles as they are extruded, and the ability to rotate the profiles as they are extruded. These options allow you to create some interesting surfaces, as will be demonstrated in the examples in this section.

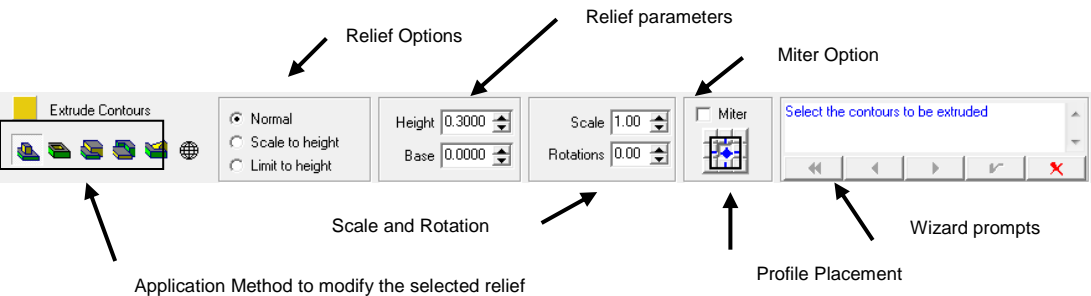
The image below shows an extrusion where the profile was rotated as it was extruded.



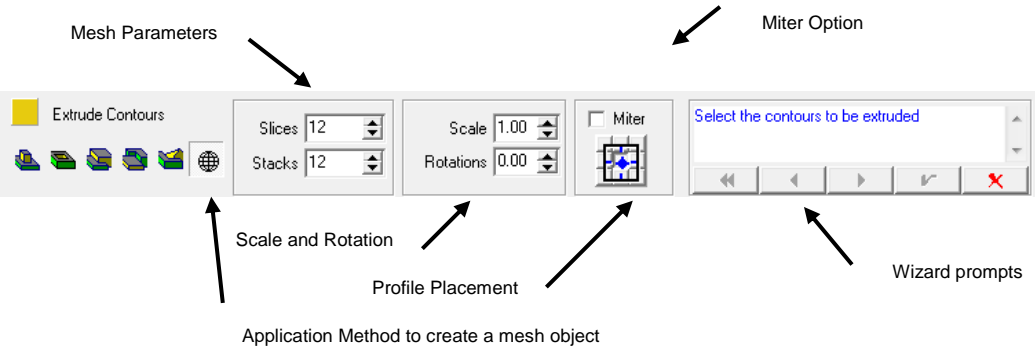
Extrude Dialog

Selecting the **Extrude** icon , or selecting the **Surface** menu and then **Create Surface** and **Extrude**, will activate the Extrude dialog. This dialog allows you to select the appropriate options and parameters required to create the new surface.

The Extrude dialog provides two sets of parameters depending on whether you have selected the option of creating a mesh object, or any other application method for modifying an existing relief. If you select an application method for modifying a relief, the following dialog is presented.



If you select the **Application Method** for creating a mesh object, the dialog is changed to allow you to enter the proper parameters for the mesh, and shown below.




Application Method

All of the options for applying the results of the extrude operation are the same as the options described in the chapter on relief creation. The options for Add, Subtract, Merge Highest, Merge



Lowest, and Replace all apply to the situation where the revolution is to be applied to an existing relief. For this to be possible, it is necessary to select the relief to be modified before starting the Extrude command.

The right-most application method option  allows you to create a mesh object with the Extrude command. As shown in the dialogs above, when this option is selected, the parameters required for the command are different than when a relief is to be modified. When this option is selected, the parameters change, providing space to enter the number of slices and stacks that will make up the new object. These are described below, and the rest of the Extrude command is the same whether you are modifying a relief or creating a new mesh object.

Relief Options

These options function similar to how they work in the relief creation tool. In this case you have three options, Normal, Scale to Height, and Limit to Height.

- **Normal** – The new extruded surface is created just as the selected profile and other options define it, without any vertical scaling.
- **Scale to Height** – The new extruded surface is scaled to match the height parameter as it is applied to the relief. This allows you to create a surface and scale it in one step.
- **Limit to Height** – The new extruded surface is limited in height by the height parameter. If a portion of the new surface extends above this height, it is truncated before it is applied to the selected relief.

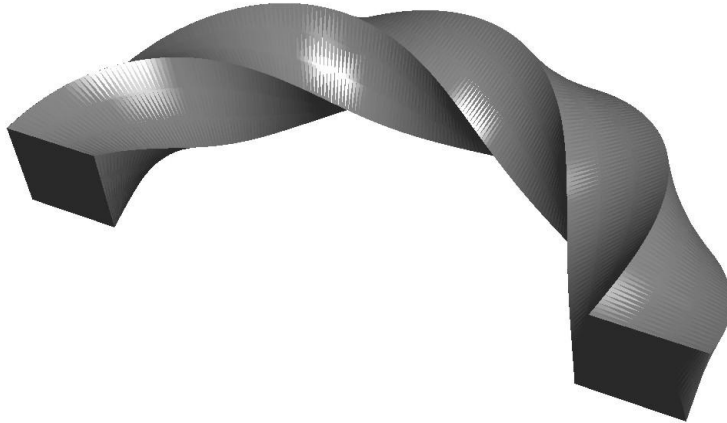
Scale and Rotation

These options allow you to modify the extruded surface. The **Scale** factor identifies the proportion that the profile is to be scaled along the length that it is extruded. If you don't want to scale the profile, then define a value of 1.0. If you want to reduce the profile size by one half, then define a value of 0.5 for the Scale factor.

The following image shows an object created using a circle extruded along the length of a semi-circle. The scale factor for this extrusion was 0.25.



The **Rotation** parameter defines the number of times that the profile will be rotated as it is extruded. A value of 1.0 will cause the profile to be rotated 360 degrees over the length of the extrusion. This is illustrated in the following graphic that was created by extruding a square profile along a semi-circle while rotating it one time.

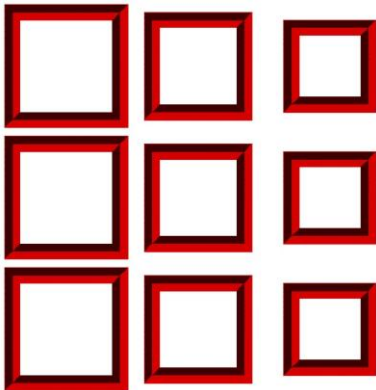


Profile Placement



The Profile Placement buttons allow you to position the extruded profile in nine different positions relative to the path. Each button indicates a position on the selection of profiles that will be extruded. This position is the location that the profiles will be placed as they are extruded around the path.

If you select the lower left button, then the lower left corner of the selected profile is where it is located as it is extruded around the path. The following graphic shows a top view of set of simple extrusions that were each created using the same profile and the same path, with just the Profile Placement button changing for each object. The top view and right view show how the objects were created both horizontally and vertically.



Top View


Right View

Relief Parameters

The relief parameters of **Height** and **Base** are available when modifying a relief.

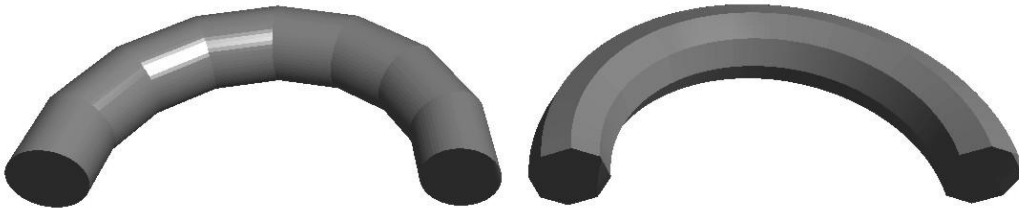
- **Height** – This value defines the height if either the ‘Scale to Height’ or ‘Limit to Height’ relief option is selected.
- **Base** – This value allows you to define a base height that can be added to the extruded surface as it is applied to the selected relief.

Mesh Parameter

When the mesh option  is selected, it activates parameters that are specific to this option. They include the number of **Slices** and the number of **Stacks** that are to be used to create the mesh object.

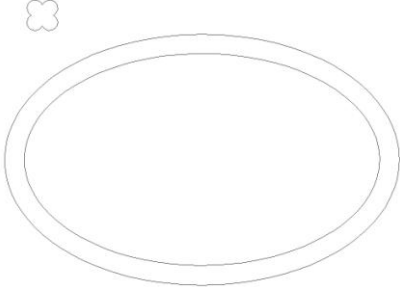
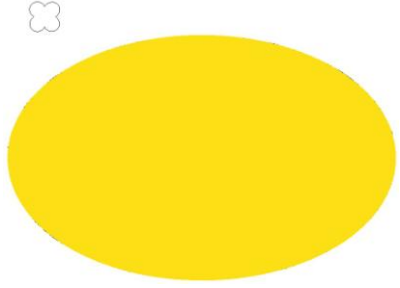



A mesh object is composed of triangles, or facets, that make up its surface. It is necessary to instruct EnRoute regarding how many of these facets should be used to construct the object. The number of **Slices** is the number of sections that are used along the length of the profile that is spun. The **Stacks** value is the number of sections that are used along the extrusion.


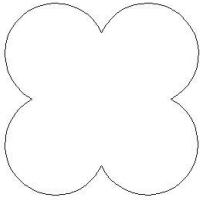
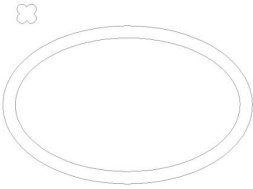

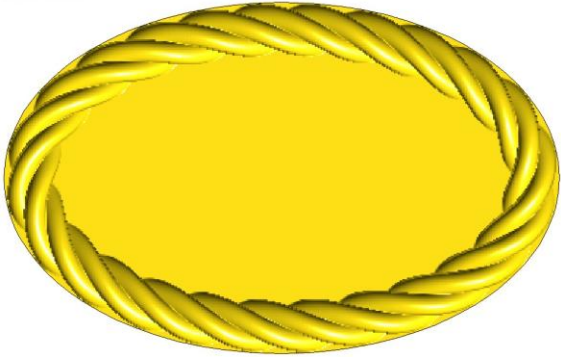
The following graphic illustrates this concept. The same profile was used to create these objects. The object on the left was created using more slices and fewer stacks, and the object on the right was created using fewer slices and more stacks.

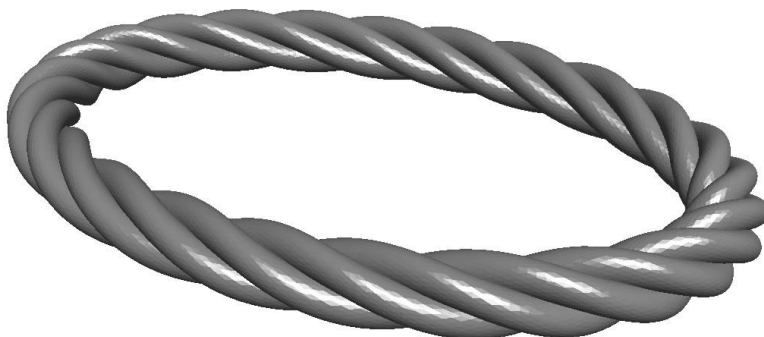


Modify a Relief Surface

This section provides a table with the steps necessary to create an extruded surface using the option of modifying an existing relief.

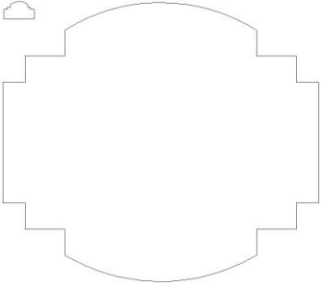






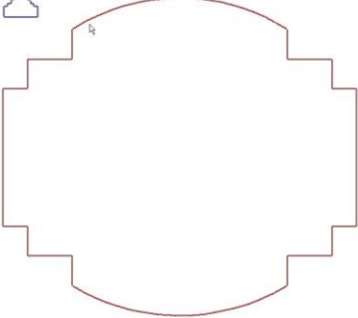
1. Create the 2D artwork. In this example, create an ellipse and then outline it. Then place four overlapping circles and weld them together. After they are welded, if there is a center hole, ungroup the object and delete the center.	
2. Select the ellipse and use it to create a flat relief.	
3. Select the relief, and then select the Extrude icon, or select the Surface menu and then Create Surface/Extrude .	
4. Click on the Add icon to select that option.	
5. Select the Normal application method.	
6. Define Scale = 1.00 and Rotations = 0.0.	
7. Make sure that the Miter option is unchecked, and click on the center Profile Placement button.	


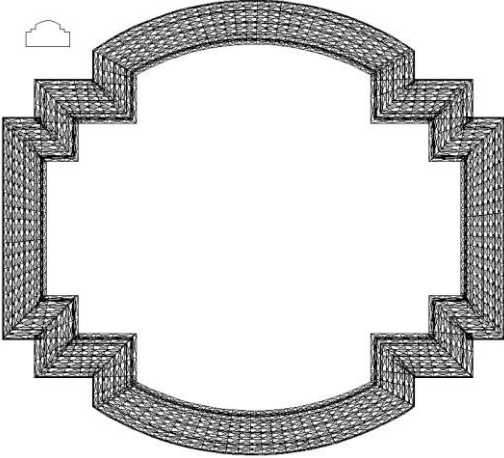
<p>8. Click to select the profile contour you want to extrude, and then click the Next Step icon  in the wizard.</p>	
<p>9. Click on the original ellipse to use as the path for the extrusion.</p>	
<p>10. Click on the Execute button to complete the command.</p>	
<p>11. The following shows a rendered top view of the resulting relief.</p>	

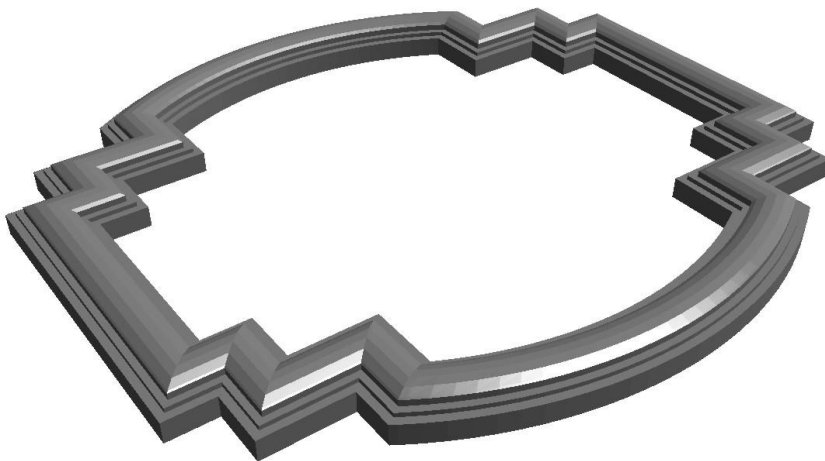


Create a Mesh Surface

The example in this section demonstrates how to create a mesh object by extruding a profile.

<p>1. Create the 2D artwork. Create a small profile as shown in the upper left, and then create a path similar to the one shown. It is not necessary to duplicate this shape precisely. It was created by drawing simple rectangles and then welding them together.</p>	
<p>2. Select the Extrude  icon, or select the Surface menu and then Create Surface/Extrude.</p>	
<p>3. Click on the Mesh icon to select that option.</p>	
<p>4. Define Slices = 24 and Stacks = 150.</p>	
<p>5. Define Scale = 1.00 and Rotations = 6.0.</p>	
<p>6. Make sure that the Miter option is checked, and click on the center Profile Placement button.</p>	
<p>7. Click to select the profile contour you want to extrude, and then click the Next Step icon  in the wizard.</p>	
<p>8. Click on the original ellipse to use as the path for the extrusion.</p>	

<p>9. Click on the Execute button to complete the command.</p>	
<p>10. The following shows a top view of the resulting object.</p>	



Sweep Two Rails

Menu: Surface / Create Surfaces / Sweep Two Rails

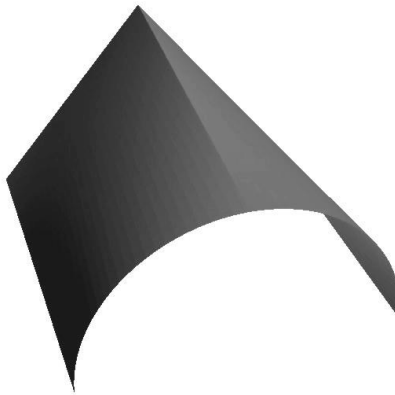
Toolbar: 3D Surfaces / Revolve / Sweep Two Rails



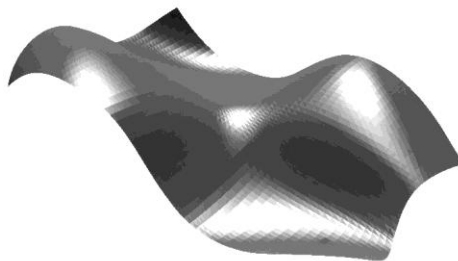
The Sweep Two Rails tool provides another method of using profile contours to either modify an existing relief or create a new mesh surface. The rails are open contours that provide a path for the right and left edges of cross section contours to follow as the profiles are used to create an extruded surface.

This tool is very flexible, and can be used to create a wide range of surfaces. It is also very useful when used to modify existing relief surfaces, particularly when the relief surface acts as a stencil to extract the swept surface. This will be illustrated by the examples later in this section.


The following simple mesh surface demonstrates the result of sweeping a semi-circle and an angled contour along two straight rails. This demonstrates how EnRoute transitions between cross sections along the length of the rails.



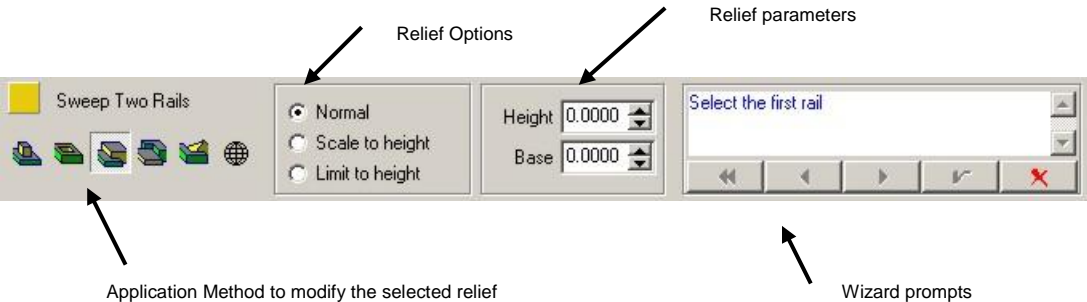
This next image demonstrates a more complex surface that was created by using curved rails and curved cross sections.



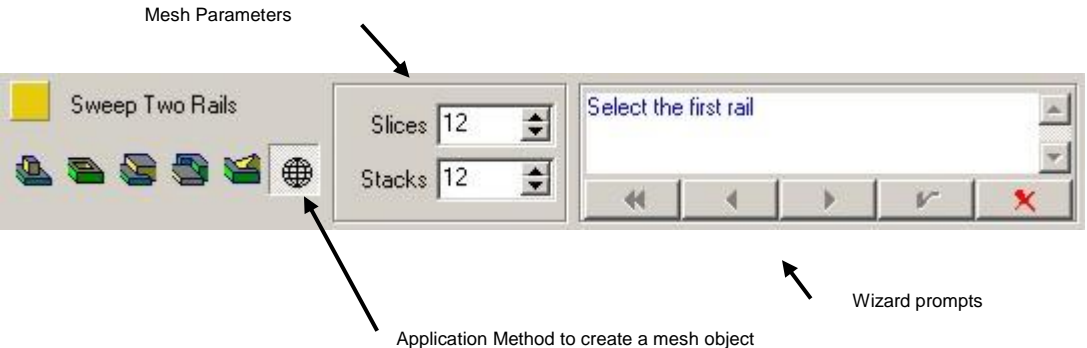
Sweep Two Rails Dialog

Selecting the **Sweep Two Rails** icon  , or selecting the **Surface** menu and then **Create Surface** and **Sweep Two Rails**, will activate the Sweep Two Rails dialog. This dialog allows you to select the appropriate options and parameters required to create the new surface.

The Sweep Two Rails dialog provides two sets of parameters depending on whether you have selected the option of creating a mesh object, or any other application method for modifying an existing relief. If you select an application method for modifying a relief, the following dialog is presented.



If you select the mesh application method, then the proper parameters for creating a mesh object are presented in the following dialog.




Application Method

All of the options for applying the results of the sweep two rails operation are the same as the options described in the chapter on relief creation. The options for Add, Subtract, Merge Highest, Merge



Lowest, and Replace all apply to the situation where the swept surface is to be applied to an existing relief. For this to be possible, it is necessary to select the relief to be modified before starting the Sweep Two Rails command.

The right-most application method option  allows you to create a mesh object with the Sweep Two Rails command. As shown in the dialogs above, when this option is selected, the parameters required for the command are different than when a relief is to be modified. When this option is selected, the parameters change, providing space to enter the number of slices and stacks that will make up the new object. These are described below, and the rest of the command is the same whether you are modifying a relief or creating a new mesh object.

Relief Options

These options function similar to how they work in the relief creation tool. In this case you have three options, Normal, Scale to Height, and Limit to Height.


- **Normal** – The new surface is created just as the selected profile and other options define it, without any vertical scaling.
- **Scale to Height** – The new surface is scaled to match the height parameter as it is applied to the relief. This allows you to create a surface and scale it in one step.
- **Limit to Height** – The new surface is limited in height by the height parameter. If a portion of the new surface extends above this height, it is truncated before it is applied to the selected relief.

Relief Parameters

The relief parameters of **Height** and **Base** are available when modifying a relief.

- **Height** – This value defines the height if either the ‘Scale to Height’ or ‘Limit to Height’ relief option is selected.
- **Base** – This value allows you to define a base height that can be added to the extruded surface as it is applied to the selected relief.

Mesh Parameters

When the mesh option  is selected, it activates parameters that are specific to this option. They include the number of **Slices** and the number of **Stacks** that are to be used to create the mesh object.

A mesh object is composed of triangles, or facets, that make up its surface. It is necessary to instruct EnRoute regarding how many of these facets should be used to construct the object. The number of **Slices** is the number of sections that are used along the width of the cross sections. The **Stacks** value is the number of sections that are used along the rails.

Wizard Prompts

The Sweep Two Rails tool requires more than one step in order to complete it. This portion of the dialog contains an area that contains a prompt that tells you the next step required in order to complete the function. It also contains buttons that allow you to move from step to step, back up a step, back up to the start of the function, execute the function, or exit the function.




The following table lists the prompts in the **Sweep Two Rails** tool, along with an explanation of the appropriate action.

Select the first rail	Click on the first open contour you would like to use as a rail.
Select the second rail	Click on the second open contour you would like to use as a rail.
Select the sweep sections	Click on the contours you want to use as the cross sections for the surface. EnRoute will draw a green line between the rails for each section you pick.

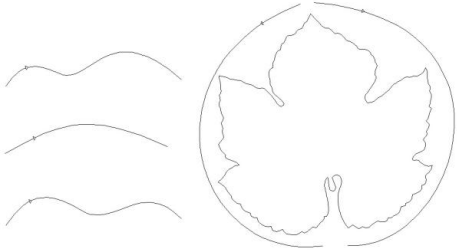
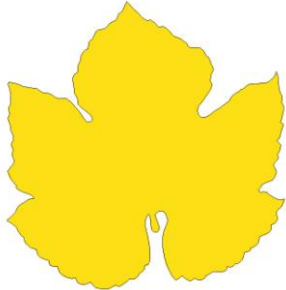





Following is a listing of each of the wizard buttons and its function.



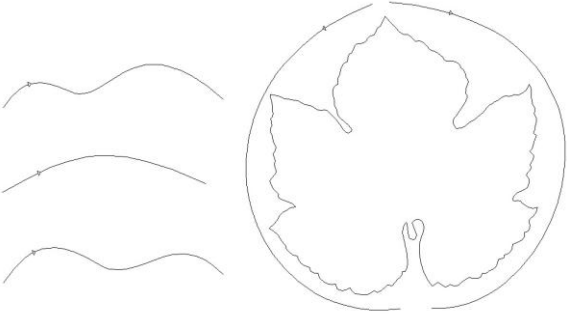



	Start	Return to the start of the command
	Back	Go back one step
	Next	Go to the next step
	Execute	Execute the function
	Cancel	Cancel the function

 You can use a contour more than once as a cross section. Also, remember that you can click and drag on the green lines to move the cross sections after they have been selected.

Modify a Relief

The steps in this section demonstrate how to use the Sweep Two Rails tool to modify an existing relief. In this case we will start with a flat relief in the shape of a grape leaf, and then use the Sweep Two Rails function as a way to create a more realistic surface shape for the leaf.

<p>1. First create the 2D artwork. In this case, that includes a contour for the shape of the leaf, contours for the cross sections, and two open contours for the rails. The contours used for this example are shown to the right.</p>	
<p>2. Select the leaf contour and use it to create a flat relief.</p>	
<p>3. Select the relief, and then select the Sweep Two Rails  icon, or select the Surface menu and then Create Surface/Sweep Two Rails.</p>	
<p>4. Click on the Add icon to select that option.</p>	
<p>5. Select the Normal application method.</p>	
<p>6. Define Height = 0.00 and Base = 0.0.</p>	
<p>7. The wizard is prompting to select the first rail. Click on the left rail contour and then click the Next step icon  in the wizard.</p>	

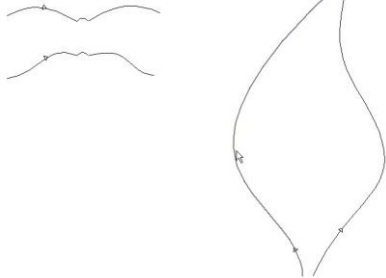




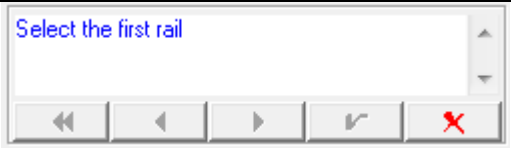
<p>8. The wizard will prompt to select the second rail. Click on the right rail contour and then click the Next Step icon  in the wizard.</p>	
<p>9. The wizard will prompt to select the cross sections. Click on the cross section contours. Each time, EnRoute will draw a green line to show where the contour was placed on the rails. The first two you identify will be placed at each end, and subsequent sections will be dividing the length.</p>	
<p>10. You can click and drag on the green lines to move cross sections along the rails to the desired position.</p>	
<p>11. Click on the Execute button to complete the command.</p>	
<p>12. The following shows a rendered top view of the resulting relief.</p>	


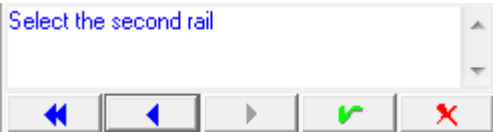
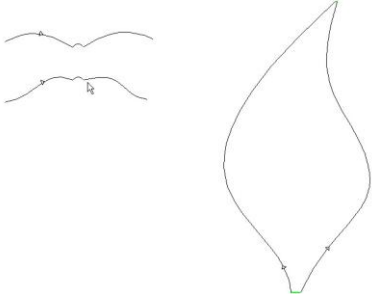

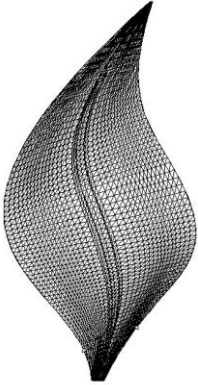
This example demonstrates how the Sweep Two Rails command can be used to modify a surface to create an organic shape. In this example, with just a few more steps, the leaf surface can be completed as shown in the following image.

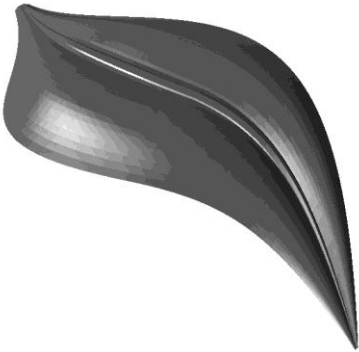


Create a Mesh Surface

The **Sweep Two Rails** tool can also be used effectively to create interesting mesh surfaces. In the following example, we create another leaf shape using a few simple contours.

<ol style="list-style-type: none"> 1. Create the 2D artwork. This example requires four contours; two for the rails, and two for the cross sections. In this example, the two rail contours have been edited so that they have dimension in the z-axis rather than being flat. 	
<ol style="list-style-type: none"> 2. Select the Sweep Two Rails  icon, or select the Surface menu and then Create Surface/Sweep Two Rails. 	
<ol style="list-style-type: none"> 3. Click on the Mesh icon to select that option. 	
<ol style="list-style-type: none"> 4. Define Slices = 36 and Stacks = 72. 	
<ol style="list-style-type: none"> 5. The wizard is prompting to select the first rail. Click on the left rail contour and then click the Next step icon  in the wizard. 	

<p>6. The wizard will prompt to select the second rail. Click on the right rail contour and then click the Next Step icon  in the wizard.</p>	
<p>7. The wizard will prompt to select the cross sections. Click on the cross section contours. Each time, EnRoute will draw a green line to show where the contour was placed on the rails. The first two you identify will be placed at each end, and subsequent sections will be dividing the length.</p>	
<p>8. Click on the Execute button to complete the command.</p>	
<p>9. The following shows a top view of the resulting object.</p>	



15. Chamfering

Introduction

The Chamfer tools allow you to modify reliefs by applying profiles that can range anywhere from a simple rounded edge all the way to an intricate profile that changes the whole relief.

Because of the way that this tool works, it is used only to modify reliefs rather than to create mesh objects. The operation of the Chamfer tools is very similar to the other relief tools.

The following image shows an example of the Chamfer Centerline tool that was used to create this intricate surface, using just one simple profile. In this case, the profile was applied to selected contours and added to the elliptical relief that already had a rounded surface.



Standard Chamfer

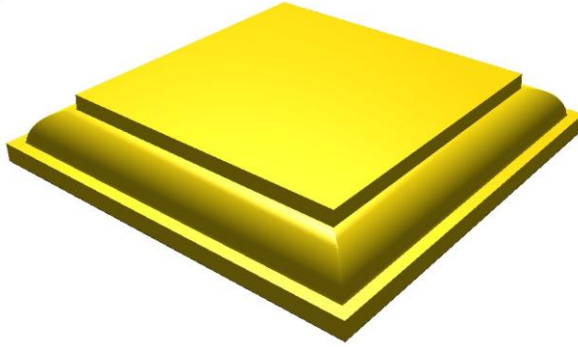
Menu: Surface / Chamfer / Chamfer

Toolbar: 3D Surfaces / Chamfer Centerline / Chamfer

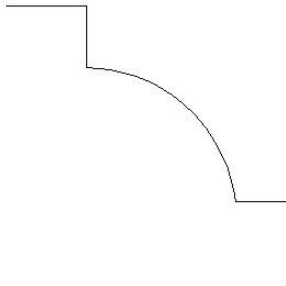


The Standard Chamfer works by first adding a height to a relief that is equal to the height of the profile, and then removing material around the perimeter of the relief that is the shape of the profile.

An analogy for this method would be to visualize the result created by passing a hand-held router with a shaper tool around the edge of a block of wood. The following image illustrates how this might look using a common bit shape.

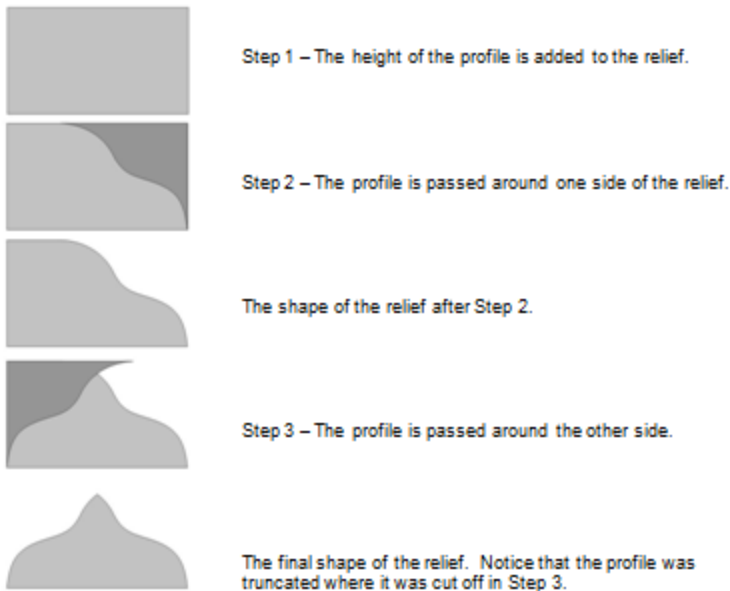


Before using the Chamfer tool, this relief was a flat relief with zero height. The shape of the profile used to create the relief is shown below. This profile, which is just an open contour in EnRoute, demonstrates the general shape, with the high point on the left side and the low point on the right side that is used for most chamfer operations.

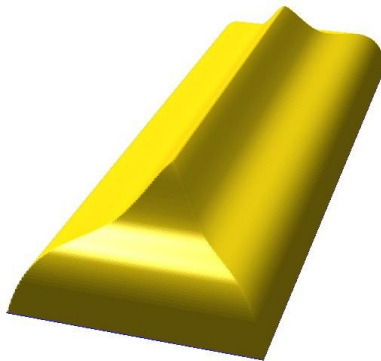


Notice that in the corners, the profile is cut off very similar to what would happen in the hand-held router analogy.


As long as the relief is wider than the chamfer profile the full width of the profile is applied. As the profile is passed around the perimeter of the relief (or the perimeter of the selected closed contours), it is possible that one side might 'cut off' the other side leaving only a partial profile. This can create an attractive effect, or you may see that you need to modify your profile to keep this from happening. The following graphic illustrates how this works.




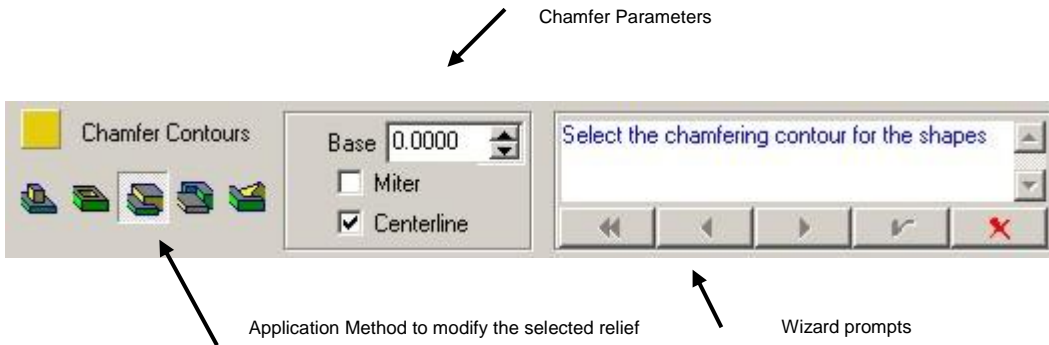
The following image shows how this looks with a relief in EnRoute.



Standard Chamfer Dialog


Activate the **Chamfer** dialog by clicking on the **Chamfer** icon , or by selecting the **Surface** menu item and then **Chamfer/Chamfer**. Prior to activating the command, select the relief you want edit. If you want to use separate contours to create the chamfered surface and then have this applied to the selected relief, then you also need to select these contours before activating the **Chamfer** dialog.

-  You can edit more than one relief with a chamfer operation just by selecting all of the reliefs you want to edit before you start the command.



Application Method

All of the options for applying the results of this operation are the same as the options described in the chapter on relief creation. The options for Add, Subtract, Merge Highest, Merge Lowest, and

Replace  all apply to the selected relief or reliefs. If you have selected contours in addition to the relief, they will be used to create the chamfer, otherwise the boundary contours of the relief will be used for the chamfer.

Chamfer Parameters

It is only necessary to specify two parameters for the chamfer command, but depending on the type of relief you are editing, your choices can have a dramatic effect on the outcome.

Base

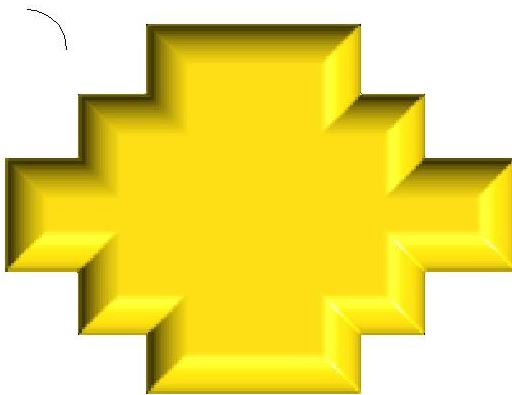
The **Base** parameter allows you to specify an additional relief height that will be added to the height of the profile contour when the chamfer is performed. The effect of this parameter is the same as with other relief creation commands.

Miter and Centerline

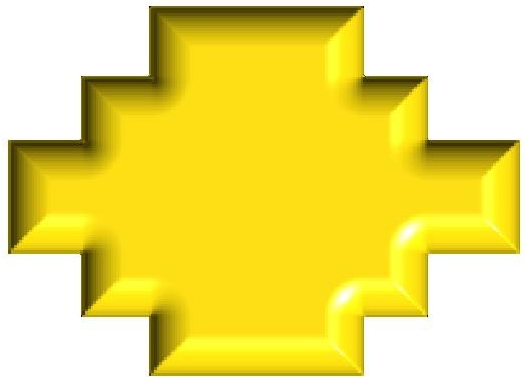
The **Miter** and **Centerline** specifications allow you to choose how the chamfer tool applies the profile in corners. For simple shapes and simple profile shapes, you likely won't notice much difference, but as the shape of the relief becomes more complex, and with different profiles, this parameter becomes important.

When specifying this parameter, you have the choice of selecting **Miter** or **Centerline**, or you have the option of using neither.

The best way to explain the effect of this parameter is to show a few examples. The following image demonstrates the difference between selecting Miter or No Miter using a simple rounded profile for the chamfer.



With Miter

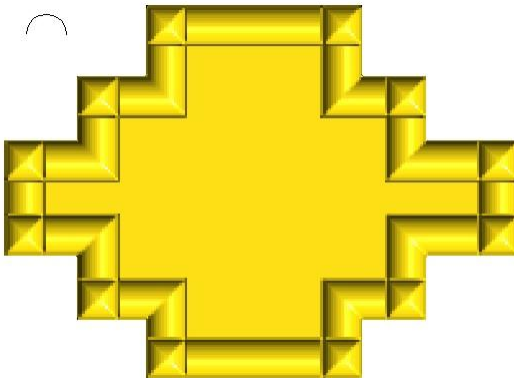


Without Miter

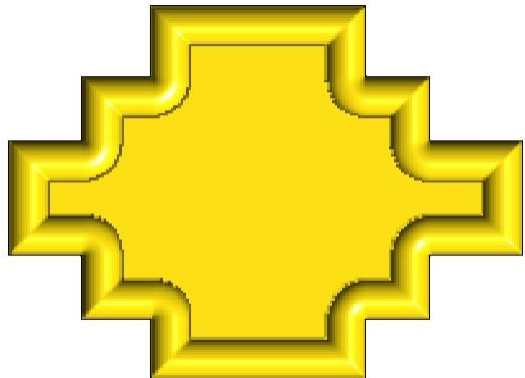
There are times when it may be appropriate to use a profile that does not adhere to the convention of placing the high side of the profile on the left edge of the profile. In these cases, the **Centerline** parameter choice will most likely be the appropriate parameter to use. The problem is that with the standard **Miter** option, the profile has the problem of cutting off the relief in the corners, creating a result that probably won't be correct.

The **Centerline** option tells EnRoute that it is necessary to maintain an imaginary bisector of the corner angle, and not allow the profile to cut itself off as it moves into and out of the corner.

Again, an illustration provides the best illustration of this issue. In the following image, you can see that the relief on the left has the problem of getting cut off by the profile as it moves around the outside corners. Fortunately, the simple solution is to choose the **Centerline** option.



Miter Option



Centerline Option

The **Centerline** option allows you to utilize profiles that are even more complex. The following example shows how to use a profile that replicates a raised-panel shape that is common in the cabinet-making industry.

The profile shape is shown below. The wide flat section on the left side of the profile is meant to

ensure that it extends across the width of the contours.



This profile was used to create the following relief. A rounded profile was also used around the outside of the relief.



Wizard Prompts




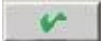

The Chamfer tool allows you to complete the command in two steps, with the second step being options. The wizard portion of the dialog contains an area that contains a prompt for the next step required in order to complete the function. It also contains buttons that allow you to move from step to step, back up a step, back to the start of the function, execute the function, or exit the function.



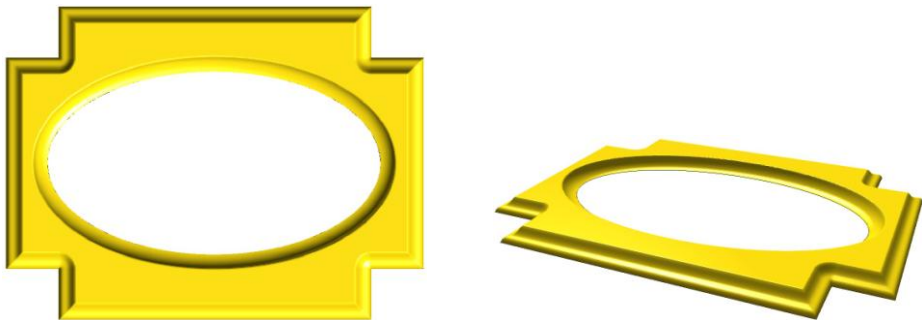
The following table lists the prompts in the **Sweep Two Rails** tool, along with an explanation of the appropriate action.

Select the chamfering contour for the shapes	Click on the open contour you want to use as the profile contour for the chamfer.
Select the chamfering contour for the holes (Optional)	Click on the contour you want to use to chamfer the holes in your relief or the holes in the selected contours. This step is optional and the profile contour selected in the first step will be used for both containers and holes by default.

Following is a listing of each of the wizard buttons and its function.

	Start	Return to the start of the command.
	Back	Go back one step.
	Next	Go to the next step.
	Execute	Execute the function.
	Cancel	Cancel the function.

You will notice that the optional second step in the wizard is to identify a different profile contour to use for chamfering the holes in your selection. While this option will likely not be used too often, the following images show how it can be used effectively to create an interesting surface in one process.



Chamfer Centerline

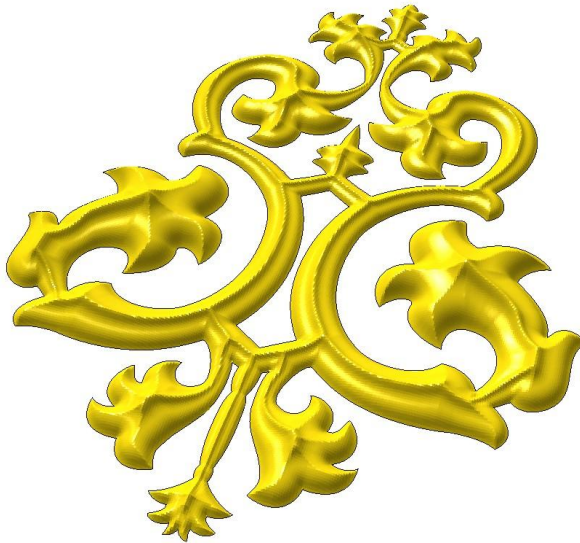
Menu: Surface / Chamfer / Chamfer Centerline

Toolbar: 3D Surfaces / Chamfer Centerline



The **Chamfer Centerline** tool provides a very useful variation on the standard chamfer tool. There are two key aspects of how it works that are different from the standard **Chamfer** tool. First, the profile contour is extended to the center of the relief shape being chamfered; second, the profile contour is scale as necessary to create the chamfer surface. This makes the **Chamfer Centerline** tool less mechanical, and allows you to create very natural-looking surfaces.

The following surface was created in one step using just the **Chamfer Centerline** tool.



One thing to note in this image is that EnRoute automatically adjusts the profile as the relief goes into corners, creating a very natural-looking surface. This does not require any input on the part of the user.

The following surfaces were created using the Standard Chamfer and Chamfer Centerline tools. You can see that the surface on the left provides a smooth, very pleasing surface.



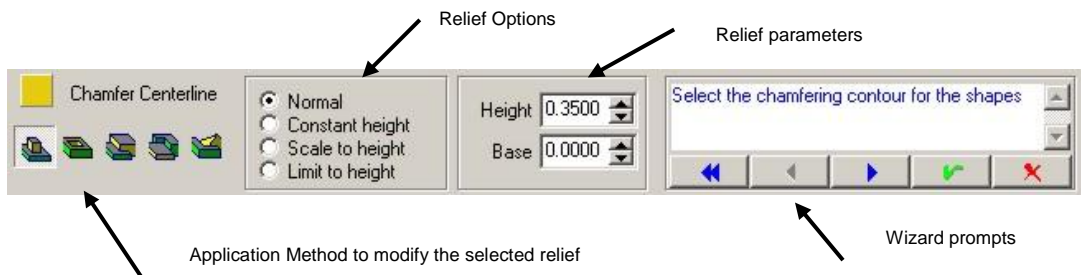
Standard Chamfer



Chamfer Centerline

Chamfer Centerline Dialog

The dialog for the **Chamfer Centerline** tool is very similar to the **Standard Chamfer** dialog. The wizard works identically, as well as the relief application options. The only differences are that the Relief Parameters are a little different, and you have the Relief Options that provide more relief height options. Those differences will be explained in this section. Please refer to the **Standard Chamfer Dialog** section for other explanations.



Relief Options

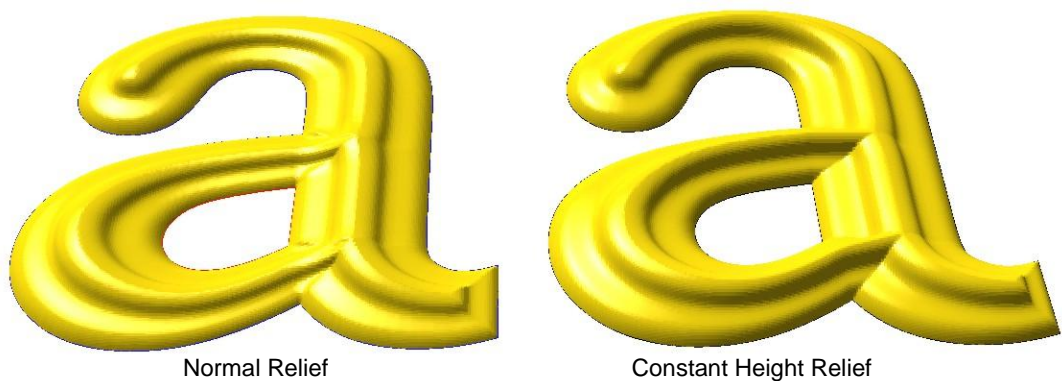
These options function similar to how they work in the relief creation tool. In this case you have three options, Normal, Constant Height, Scale to Height, and Limit to Height.

- **Normal** – The new surface is created just as the selected profile and Relief Parameters define it, without any additional vertical scaling.
- **Constant Height** – The new surface is created to match the height specified in the **Height** parameter.

- **Scale to Height** – The new surface is scaled to match the Height parameter as it is applied to the relief. This allows you to create a surface and scale it in one step.
- **Limit to Height** – The new surface is limited in height by the height parameter. If a portion of the new surface extends above this height, it is truncated before it is applied to the selected relief.

📄 With the **Scale to Height** option, the relief is first created using the **Normal** option, and then it is scaled so that the maximum height of the relief matches the **Height** parameter.

The **Constant Height** option works quite well with the Chamfer Centerline tool. This is worth noting because the results you get by selecting the Constant Height option are often quite improved over the results without it. The following image demonstrates the different results that can be obtained by choosing this option.



Relief Parameters

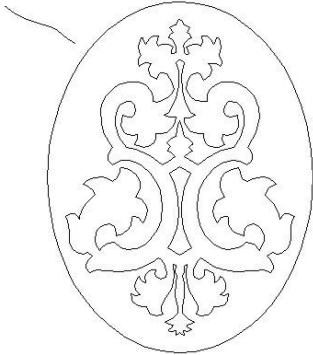
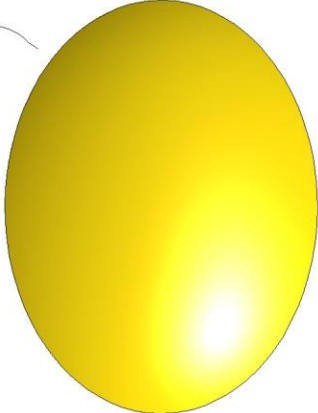


As with all the other tools, the **Base** parameter provides a means of adding on to the relief height to create vertical sides on the relief.

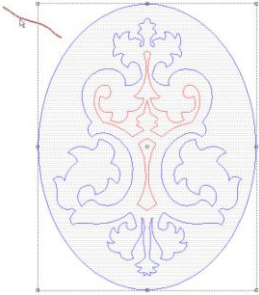


The **Height** parameter will be used every time the Chamfer Centerline tool is used. The following table shows how the **Height** parameter is used with each of the **Relief Options**.

Normal	Height of the tallest point in the relief.
Constant Height	Height of the relief in general.
Scale to Height	Height of the tallest point in the relief. In this case, it gives the same results as the Normal option.
Limit to Height	Maximum height of the relief.

Chamfer Centerline Example

The following example shows the steps involved in creating a relief surface using the Chamfer Centerline tool.

<p>1. First create the 2D artwork. In this case, that includes an ellipse that is approximately 7 inches wide by 9 inches tall. The primary artwork includes some fancy 2D clipart that is commercially available. The open contour will be the profile for the chamfer. It is a simple Bezier curve drawn in the EnRoute.</p>	
<p>2. Select the ellipse contour and create a 100 dpi rounded relief using an angle of 30 degrees.</p>	
<p>3. Select the relief and the clipart, and then select the Chamfer Centerline icon , or select the Surface menu and then Chamfer/Chamfer Centerline.</p>	
<p>4. Click on the Add icon to select that option.</p>	
<p>5. Select the Normal application method.</p>	
<p>6. Define Height = 0.30 and Base = 0.0.</p>	

<p>7. The wizard is prompting to select the chamfer profile contour. Click on the open profile contour to select it.</p>	
<p>8. Click on the Execute button to complete the command.</p>	
<p>9. The following shows a rendered top view of the resulting relief.</p>	

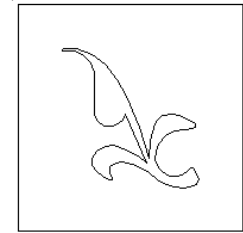





Baroque Carve Chamfer



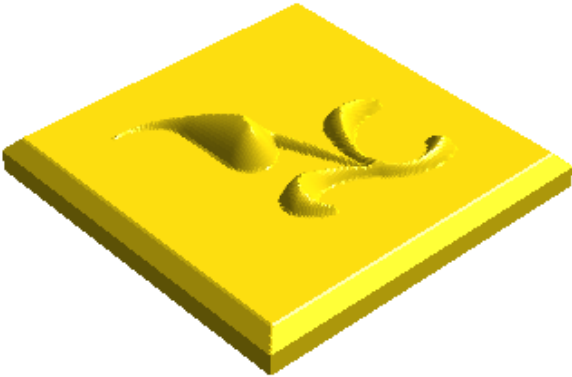
Menu: Surface / Chamfer / Baroque Carve Chamfer

Toolbar: 3D Surfaces / Chamfer Centerline / Baroque Carve Chamfer

The Baroque Carve Chamfer tool can be used to create the look of wood carving. The tool allows you to use a different profile shape for the outside and the inside of the contour.

The following example shows the steps involved in creating a relief surface using the Baroque Chamfer tool.

<p>1. First create the 2D artwork. In this case, that includes a 5 inch square. The primary artwork includes some fancy 2D clipart that is commercially available.</p>	
<p>2. The open contours will be the profile for the outside and inside chamfer. It is a simple Bezier curve drawn in EnRoute.</p>	<div data-bbox="633 412 780 500" style="display: inline-block; border: 1px solid black; padding: 2px;">Outside contour</div>  <div data-bbox="1063 412 1210 500" style="display: inline-block; border: 1px solid black; padding: 2px;">Inside contour</div>
<p>3. Select the square contour and create a 100 dpi limited height, beveled relief using an angle of 45 degrees.</p>	
<p>4. Select the relief and the clipart, and then select the Baroque Chamfer icon.</p> 	
<p>5. Click on the Add icon to select that option.</p>	
<p>6. Select the Limit Height application method.</p>	
<p>7. Define Height = 0.50 and Base = 0.0.</p>	
<p>8. The wizard is prompting to select the chamfer profile contour for the outside curves. Click on the outside open contour profile to select it. Click on the forward arrow.</p>	

<p>9. The wizard is prompting to select the chamfer profile contour for the inside curves. Click on the inside open contour profile to select it.</p>	
<p>10. Click on the Execute button to complete the command.</p>	
<p>11. The following shows a rendered perspective view of the resulting relief.</p>	

16. Using 3D Meshes

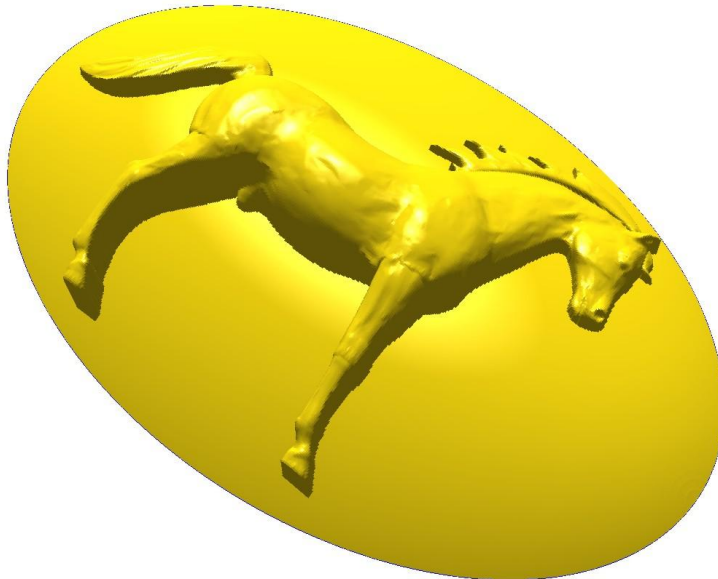
Introduction

The 3D modeling process uses 3D surfaces to define and depict the objects in the design. There are several different ways to accomplish this. In EnRoute, **reliefs** are used as the method for defining and manipulating 3D surfaces that are used to create toolpaths for output.

Another common method of defining 3D objects is as **3D meshes**. These objects are made up of triangles of varying sizes that define their surface. EnRoute enables you to both create and utilize 3D meshes as part of the relief design process.

As with all of the different methods for defining 3D surfaces, mesh objects have advantages and disadvantages. One clear advantage is that many common 3D modeling software packages provide the ability to export and import mesh objects. For this reason, meshes are probably the most common type of 3D objects. There are many companies that sell 3D objects of just about any type of object that you can imagine. This may be a good way to acquire an object to incorporate into your design that would otherwise be difficult and time-consuming to create.

In addition to being able to import and use 3D models, many of EnRoute's modeling tools provide the option of creating a mesh object rather than modifying a relief. These meshes can then be positioned, rotated and scaled in order to make them just right for incorporating into a relief design.



Using 3D Meshes from Other Applications

EnRoute currently imports 3D mesh objects in three different formats,

- **DXF** – **D**ata **eX**change **F**ormat is the format created by AutoDesk® as a standard method for bringing data into and out of AutoCAD®. It has become a standard method for exchanging data between many software packages. EnRoute imports both 2D and 3D data using this format.
- **3DS** – This is the format used by 3D Studio®. EnRoute imports 3D files saved using this format.
- **STL** – **S**Tereo **L**ithography is a standard method created for sharing 3D mesh surface data. EnRoute imports STL mesh objects.

Creating 3D Meshes with EnRoute Tools

Within EnRoute, there are a number of ways to create 3D meshes. Most of these methods are described in detail along with the tools that provide this option. Refer to the manual sections that describe the **Revolve**, **Spin**, **Extrude** and **Sweep Two Rails** functions to see how to create these types of mesh objects.

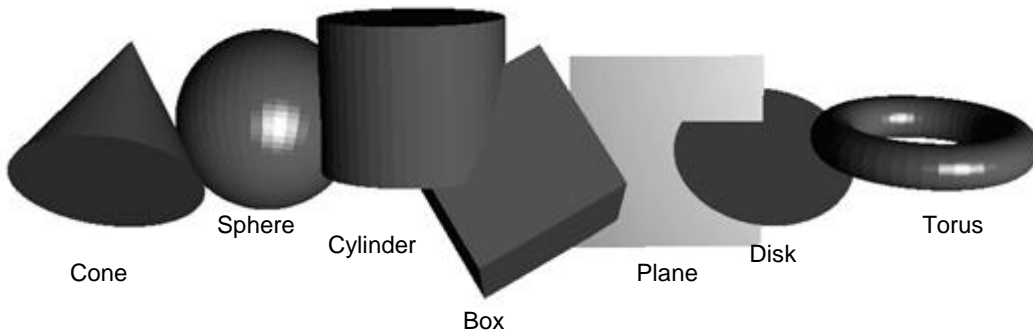
Primitives


Menu: Surface / Create Primitive

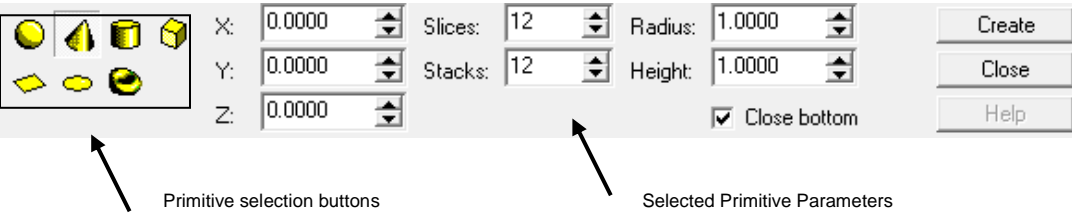
Toolbar: 3D Surfaces / Create Primitive Objects



The Primitives tool is used to create a range of basic 3D shapes. These shapes can be used to build more complex shapes, or they can be used as design elements in your relief. You likely won't need them too often, but they can come in handy.



To access the **Primitives** dialog, click the **Primitives** icon,  or select the **Surface** menu and then click **Create Primitive**. The following dialog will be activated.



Whichever primitive button is active will activate the parameters for that primitive type. The parameters are similar between primitive types, but they are specific to that type.

The following table lists the different parameter types, along with an explanation of what that parameter defines.


X,Y and Z	These are the coordinates for the location of the center of the primitive.
Slices	This is the number of sections either around the circumference of the object or along one axis.
Stacks	This is the number of sections along one axis, typically the z-axis, of the object.
Radius	This is half the diameter of a circular dimension of the object.
Height	This is the z dimension of the object.
Size	This is the dimension along one axis of the object, such as 'X Size'.
Sweep Angle	For the Torus object, this is the number of degrees of the sweep of the object around the 'outer' radius.
Close top or Close Bottom	You can choose whether Cylinders, Cones and Boxes should have surfaces that cover their tops and bottoms.


Applying Meshes to a Relief

Menu: Surface / Apply Mesh

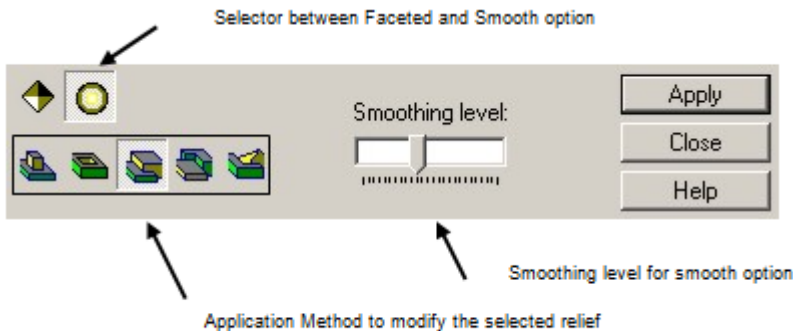
Toolbar: 3D Surfaces / Apply Mesh to Relief

After meshes are created or imported, their primary function is to be used to modify relief surfaces.

The Apply Mesh  tool is used to accomplish this.


In order to enable the **Apply Mesh** function, it is necessary to have at least one relief and one mesh in your selection. After selection a mesh and a relief, click on the **Apply Mesh** icon , or select the **Surface** menu and **Apply Mesh** to activate the dialog. To complete the function, just click on the **Apply** button.


Apply Mesh Dialog



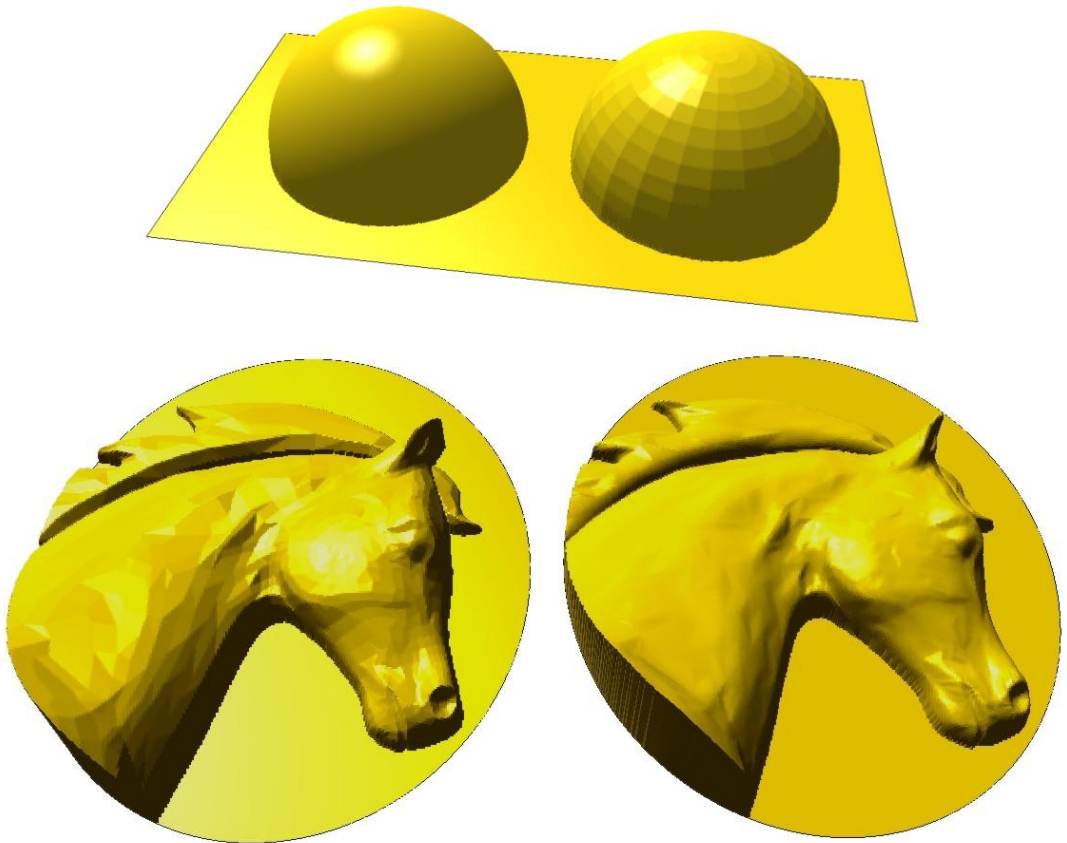
Faceted  or **Smooth** 

One of the characteristics of a mesh surface is that it is composed of triangles, or facets, that make up its surface. When a mesh object is applied to a relief surface, the facets can be quite obvious and may be an undesirable characteristic.

EnRoute provides a Smooth  option that implements a means of smoothing out the facets of the mesh to provide a smoother finished surface. For surfaces that are intended to be smooth, the

results of using this option can be quite dramatic. If the Faceted  option is selected, then the object is applied to the relief surface using its facets directly.

The following images show the difference that can be realized by using the **Smooth** option.



Application Method

All of the options for applying the results of this operation are the same as the options described in the chapter on relief creation. The options for **Add**, **Subtract**, **Merge Highest**, **Merge Lowest**, and

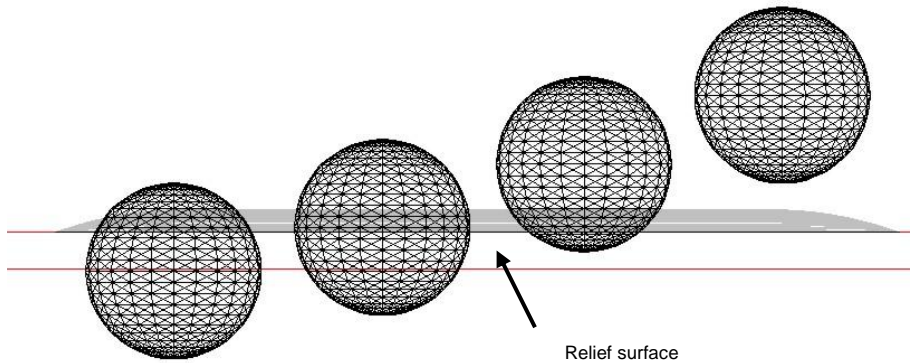


Replace

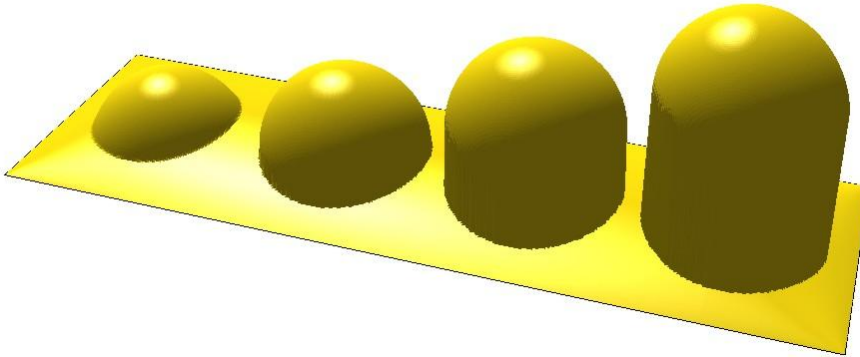
all apply to the selected relief or reliefs.

The vertical location of the mesh relative to the vertical location of the relief controls how much the relief is modified. It is important to position the mesh object correctly in the z axis relative to the relief, so when it is applied to the relief, the results are correct.

The following image shows a front view that includes a relief and four mesh spheres that are located at different positions in the z axis. A perspective view of the relief shows the results of applying these objects to the relief using the Add option.



Front View showing vertical position of meshes and relief

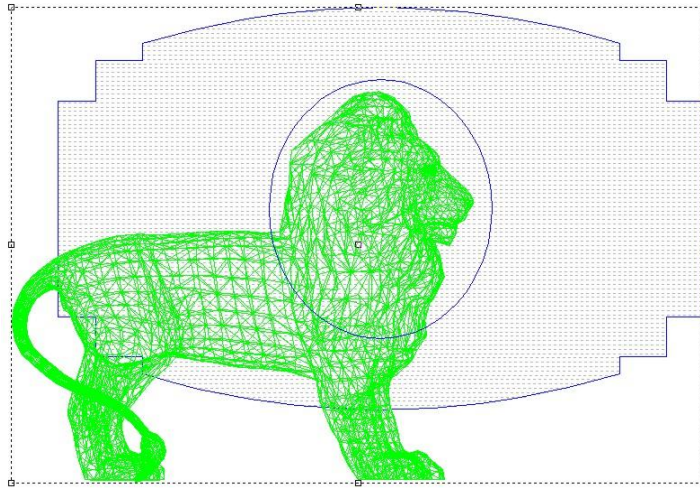


Perspective view of the relief after meshes were added

Using a Mask

There may be times when it is desirable to use just a portion of a mesh object in your design. This can be accomplished by using a mask. A mask is just a closed contour that is selected along with the relief and the mesh object when the **Apply Mesh** command is started.

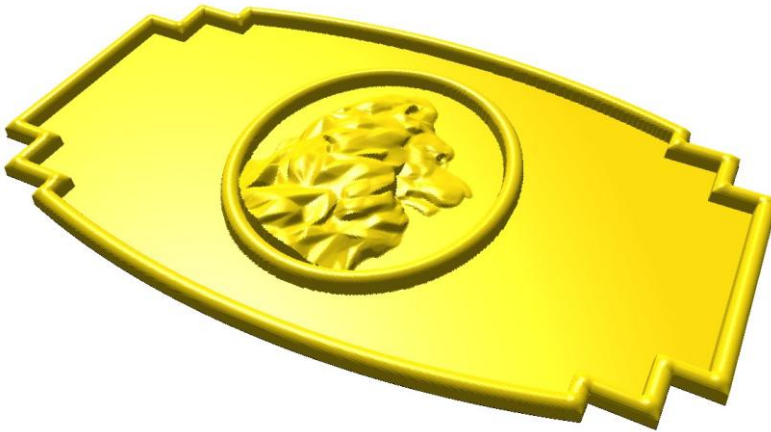
The following image shows a large relief, 3D mesh of a lion, and a smaller elliptical contour which acts as the mask. All three of these elements are part of the selection.



With these selected, execute the Apply Mesh command to add the mesh to the relief. The result is shown in the next image.



After adding some borders to the relief to complete the design, you see a perspective view of the relief in the next image. You can see how the use of the mask made it simple to incorporate only the portion of the mesh object that was needed.



17.Modifying and Combining Reliefs

Introduction

After you have created a relief, EnRoute provides many tools that allow you to modify it to fit the needs of your design. This chapter provides descriptions of the tools that are available for modifying reliefs as part of your design process.

Selecting Reliefs

To select a relief, click on the contour that defines its perimeter.

You can also select multiple reliefs using one of the following methods:



- Hold down the **SHIFT** key and click on the reliefs one after the other.
- Click and drag to draw a selection box around the reliefs.


Cutting, Copying and Pasting Reliefs

One handy aspect of reliefs in EnRoute is that they exist in the same workspace as all of the other EnRoute object types. This means that they can be treated in much the same way as any other EnRoute object.

Cutting and Pasting


To cut and paste reliefs:

1. Select the reliefs to cut.
2. From the Edit menu choose Cut, click on the Cut Icon,  or press Ctrl+X.
3. Select the drawing where the reliefs are to be pasted.
4. From the Edit menu choose Paste, click on the Paste Icon,  or press Ctrl+V.


 After a relief is placed on the clipboard either by cutting or copying it, it remains there until it is replaced by something else. This allows you to paste it more than once.

Copying and Pasting

To copy and paste reliefs:

1. Select the reliefs to copy.
2. From the Edit menu choose Copy, click on the Copy Icon,  or press Ctrl+C.
3. Select the drawing where the reliefs are to be pasted.

4. From the Edit menu choose Paste, click on the Paste Icon,  or press Ctrl+V.


 Another easy way to copy a relief is to click and drag on it while pressing the CTRL key. When you release the mouse button a copy of the selected reliefs is placed at the new location.

Deleting Relief Objects

To delete one or more reliefs:

1. Select the reliefs.


2. From the Edit menu, select Delete,  or you may also just press the DELETE key.

 Deleting relief objects is the same as deleting any other object in EnRoute.

Clearing Reliefs

Clearing a relief removes the relief surface from the object, but it retains the contours that made up the perimeter of the relief. You will use this during the relief design process to clear a relief so that you can recreate it using different parameters.

To clear a relief:

1. Select the relief.
2. From the Surface menu, select Delete Relief, or just select the Delete Relief Icon  from the relief toolbar.

Moving Reliefs

Moving reliefs is done much the same way as moving any other object in EnRoute. You can click and drag on a relief object to move it in the top, front and right views.

The key difference is that the vertical position of a relief object is very important when it comes time to create toolpaths and output toolpaths. In addition to the Precision Input Center, EnRoute provides several tools to help you position relief objects in the z-axis.

To move a relief using the Precision Input Center:

1. Select the relief in the view (Top, Front or Right) that you want to use as the reference plane. For example, in the Top view the reference plane is the X-Y plane. In the Z axis this plane is located at the closest Z-location on the object(s) you have selected.
2. Press F2 to activate the Precision Input Center.
3. Click on the Move tab.
4. Click on the button that represents the corner of the object you want to specify.



5. Specify the X, Y, and Z coordinates for the selected corner.
6. Click on the OK button to finish moving the object(s).







Vertical Positioning of Reliefs

The method for positioning reliefs described above works well, and you can use this method exclusively; however, EnRoute also provides tools that are specifically designed to assist you with positioning reliefs vertically within the boundaries of the Plate.

Access the **Align Reliefs** tools by clicking on the **Align Relief** icon in the Relief toolbar to activate the Align Reliefs flyout toolbar.



The following table describes the function of each of the Align Reliefs functions

	Align reliefs to bottom of plate	Aligns each of the selected reliefs to the bottom of the plate.
	Align selection to bottom of plate	Aligns the selection to the bottom of the plate. The relative position of the reliefs in the selection doesn't change.
	Align reliefs to top of plate	Aligns each of the selected reliefs to the top of the plate.
	Align selection to top of plate	Aligns the selection to the top of the plate. The relative position of the reliefs in the selection doesn't change.
	Align reliefs to middle of plate	Aligns each of the selected reliefs to the middle of the plate.
	Align selection to middle of plate	Aligns the selection to the middle of the plate. The relative position of the reliefs in the selection doesn't change.

Rotating Reliefs

Rotating reliefs is accomplished the same way as rotating any other objects in EnRoute. Refer to the section on **Rotating Contours** in the **Arranging Contours** chapter.

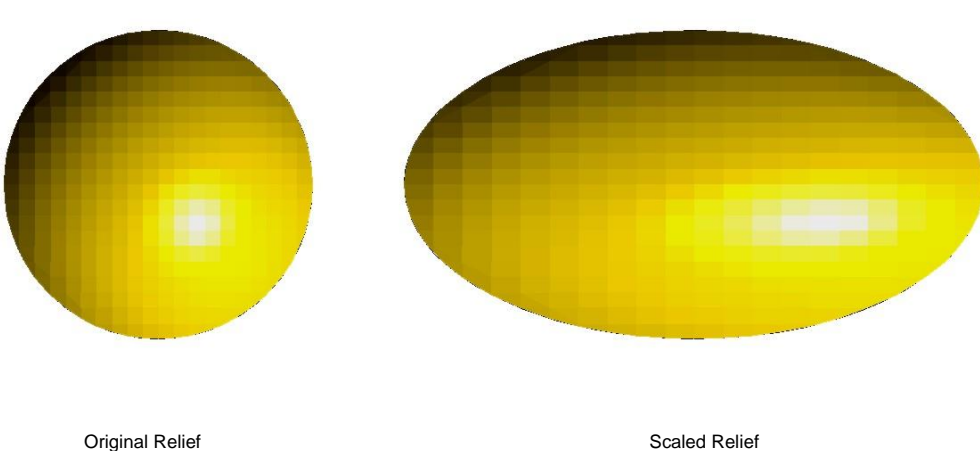
Scaling Reliefs

Scaling reliefs is accomplished the same way as scaling any other objects in EnRoute. You can either scale the relief by selecting it and then clicking and dragging on one corner of the selection box, or you can use the Precision Input Center to precisely scale the relief. (See the **Scaling Contours** description in the **Arranging Contours** chapter)

When it comes to scaling reliefs, there is an important consideration involving the **resolution** of the relief. Remember that when you first create a relief, you specify a resolution for that relief. This specifies the resolution of the grid that defines the relief surface. After the relief is created, the total numbers of grid points that define the relief are set. So, if you scale the relief, the effective size of the grid points in the relief is scaled too.

If you created a relief that was 50 mm x 50 mm with a resolution of 100 dpi and then scaled it up to a size of 500 mm x 500 mm, the effective resolution of the relief would drop to 10 dpi. This illustrates that it is important to consider the effect of scaling a relief, and the effect that it will have on the quality of the finished piece.


The following image shows how the size and shape of the relief grid is affected by a scaling operation.



Fit Relief to Plate

Menu: Surface / Fit to Plate

Toolbar: 3D Surfaces / Smooth Relief / Fit to Plate

You can use the moving and scaling tools in EnRoute to size and position a relief just as you would other objects. There are times, however, when it would be useful to be able to automatically fit a relief into your plate as you get it ready for toolpaths. The **Fit Relief to Plate**  function provides this capability. It will automatically position and scale a relief so that it fits vertically within the defined plate in your active drawing.

To fit a relief to the plate:

1. Select the relief.
2. From the Surface menu, select Fit Relief to Plate.

Smoothing Reliefs


Menu: Surface / Smooth Relief

Toolbar: 3D Surfaces / Smooth Relief

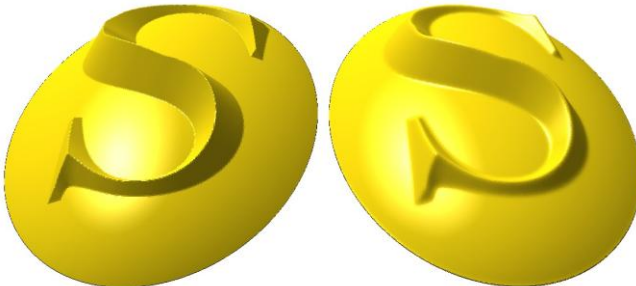
There are times when you are working on a relief design when it will be useful to be able to soften, or smooth, the relief either to make it more attractive or to make it so that it will machine better.


EnRoute provides the **Smooth Relief**  tool accomplish this.

1. Select the reliefs to be smoothed.
2. From the Surface menu, select Smooth Relief.
3. Click and drag to set the Smoothing Radius and Smoothing Power.
4. Click Apply.

 You can apply the smoothing function to a relief more than one time in order to smooth it more and more.

The following image shows an example of the results of smoothing a relief.



-  You can apply smoothing to just a portion of a relief by selecting a contour along with the relief. The selected contour(s) acts as a mask, and the relief will only be smoothed within the boundary of the contour.

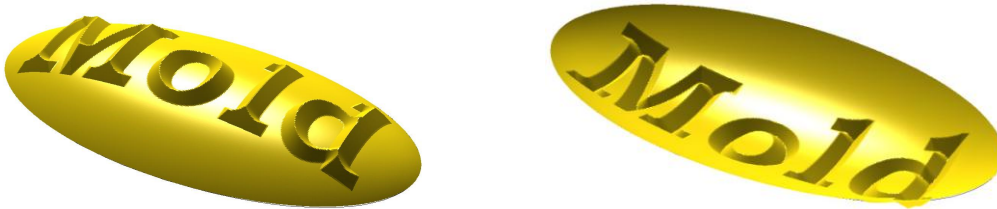
Inverting Reliefs

Menu: Surface / Invert Relief


Toolbar: 3D Surfaces / Smooth Relief / Invert Relief



Inverting a relief has the effect of reversing the z-coordinates of the relief relative to the location of the perimeter contours of the relief. This is a simple way to convert a 'positive' relief to a 'negative' relief.



To invert a relief:

1. Select the reliefs to be inverted.
2. From the Surface menu, select Invert Relief. 

You can also select the **Invert Relief** function from a screen icon. The function is located on a flyout toolbar that is activated by clicking a holding the Smooth Relief icon. The flyout is shown to the right.

One thing you can see in the images above is that inverting a relief only modifies it in the z axis. If you plan to invert a relief to use it as a mold, it may also be desirable to mirror the relief prior so that any impressions created from a mold have the proper x-y orientation. A relief can be mirrored using the same method as any other EnRoute object. From the **Transform** menu, select **Mirror** and then either **Mirror Horizontal** or **Mirror Vertical**.

The following images show the sequence of actions to both invert and mirror a relief in preparation for using it as a mold.




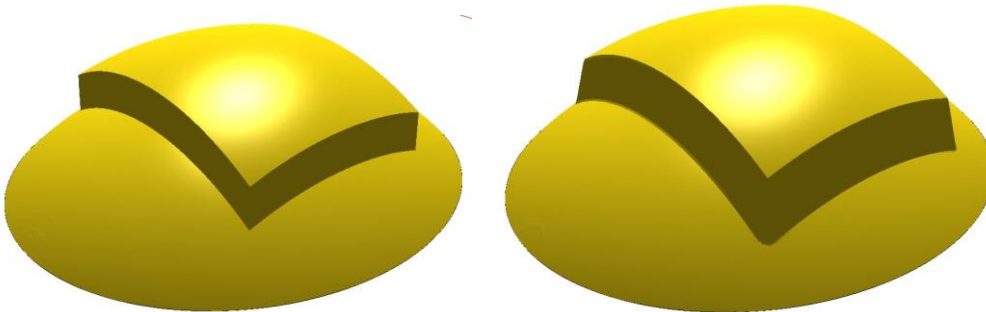
Adding a Draft Angle to a Relief

Menu: Surface / Apply Draft Angle to Relief

Toolbar: 3D Surfaces / Smooth Relief / Add Draft Angle

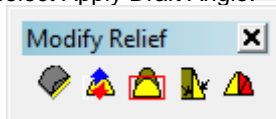
When you create a relief, you will often create portions of the relief that have a vertical side. This vertical side can be problematic for two reasons. First, if the vertical side is relatively tall, its height could be greater than the cutting length of the tool you will use to mill it with. This can cause problems that might cause the tool to break. Second, if you are going to use the finished piece to create a mold you may need to provide a draft angle in order to allow the molded pieces to come out of the mold properly.

EnRoute provides a **Draft Angle**  function that allows you to add a draft angle to your relief.



To add a draft angle:

1. Select the relief.
2. From the Surface menu, select Apply Draft Angle.  You can also access the tool through



the Modify Relief Toolbar.




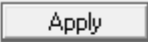
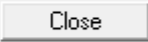


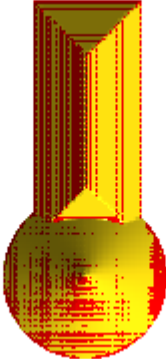

3. In the Precision Input dialog, enter desired values for Draft Angle and Height Threshold. (The draft angle is usually between 3 and 7 and the Height Threshold refers to the starting point of the angle.)
4. Click Apply.

Create a mesh surface from a Relief

Menu: Surface / Create mesh surface from relief

Toolbar: 3D Surfaces / Smooth Relief / Create a mesh surface from relief

This tool allows you to create a mesh from an existing relief.


1. Select the relief.	
2. Click and hold the Smooth icon, then click on the Create mesh from relief icon.	 Smooth Icon  Create Mesh from Relief
3. Enter the Mesh tolerance in the parameters field and click Apply.	Mesh tolerance <input type="text" value="0.0150"/>    
4. The mesh will be created and placed in the same location as the relief.	<div></div> <div>ReliefRelief and MeshMesh</div>

Merging Reliefs

Menu: Surface / Merge Reliefs

Toolbar: 3D Surfaces / Merge Reliefs

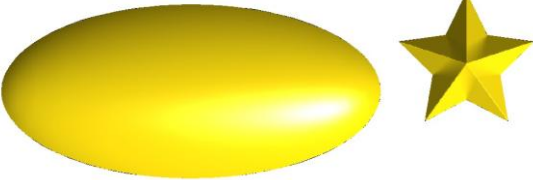
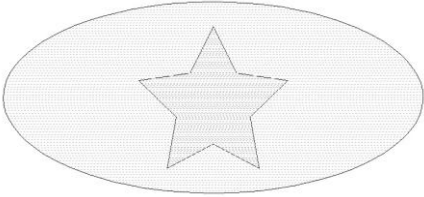

This tool allows you to merge one or more reliefs with a base relief. This allows you to create portions of a relief separately and then merge them together to create the final surface.





The following graphic shows the dialog that is activated when you select the **Merge Reliefs**  function.

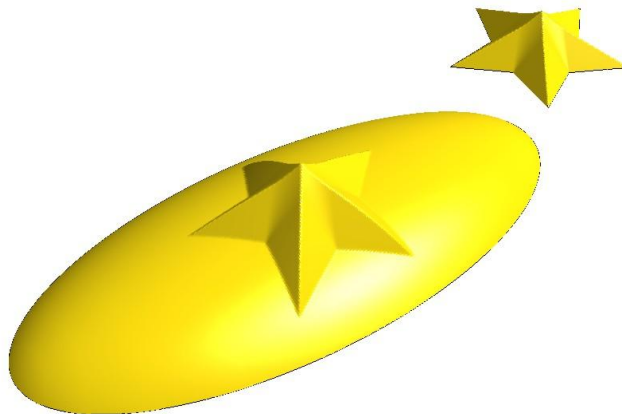


As you can see, it is not a complicated dialog. To complete the merge action, it is just necessary to select the application method for how the reliefs will be merged, select the base relief and then select the relief(s) that will be merged with the base relief.


The following example illustrates the process.

1. First create the reliefs that you will merge. In this case we have created two simple reliefs, an ellipse and a star.	
2. Position the reliefs correctly for merging.	
3. Select Merge Reliefs. 	

<p>4. Click on the Add icon to select that option in the dialog.</p>	
<p>5. The wizard prompts you to select the base relief. Click on the ellipse relief to select it, then click on the Next button.</p>	
<p>6. The wizard prompts you to select the reliefs to merge. Click on the star relief to select it, then click on the Execute button to complete the merge.</p>	
<p>7. This image shows the result of the merge action.</p>	



This image shows the merged relief along with the original star relief.

-  The Merge Relief tool can merge reliefs of different resolutions, and reliefs that have been rotated and scaled without any problems.



Combine Reliefs

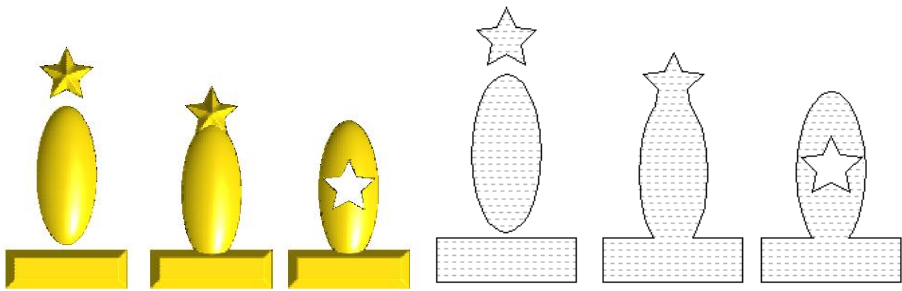
Menu: Surface / Combine Reliefs



Toolbar: 3D Surfaces / Merge Reliefs / Combine Reliefs

Use the combine Reliefs tool to combine one or more selected reliefs.

The example below demonstrates three separate reliefs combined into one relief in two different ways.

- 1. Select the reliefs to be combined.
- 2. Click and hold the Merge Reliefs Icon  to open the flyout toolbar.
- 3. Click on the Combine Reliefs Icon. 



-  If the reliefs overlap they will combine to create one relief with one contour.
-  If one of the reliefs is completely enclosed within another, it will create a hole in the relief.

Extracting Slices

Menu: Surface / Extract Slices

Toolbar: 3D Surfaces / Merge Reliefs / Extract Slices

The Extract Slices tool allows you to “cut” the relief into layers. Use this tool to “stack” your material to create large objects. For example, if your material is 1.5 inches and the object that you want to create is 4 inches, you can cut your relief into 3 slices and then stack the material to create the object.

Extract Slices Dialog

Relief Dimensions	
Height	1.3175
Top	1.3175
Bottom	0.0000

Relief Dimensions

- Height The total height of the selected relief.
- Top The location of the top of the selected relief.
- Bottom The location of the bottom of the selected relief.

Slice

Slices

3

Thickness

0.4392

Current Slice

1

Thickness

0.4392

Bottom

0.8783

☐ Create current slice

☒ Create all slices

Slice

- Slices The number of slices that you want to create.
- Thickness The thickness of each slice.
- Current Slice The current slice number. This tells you which slice parameters are displayed. The slices are numbered from top to bottom.
- Thickness Thickness of the current slice.
- Bottom Location of the bottom of the current slice.
- Create Current Slice This tells EnRoute to create only the slice that you have selected to be the current slice.
- Create all slices Select this parameter to create all of the slices.



Click this button to create the slices.





Click this button to exit the tool without creating the slices.


Example 1:

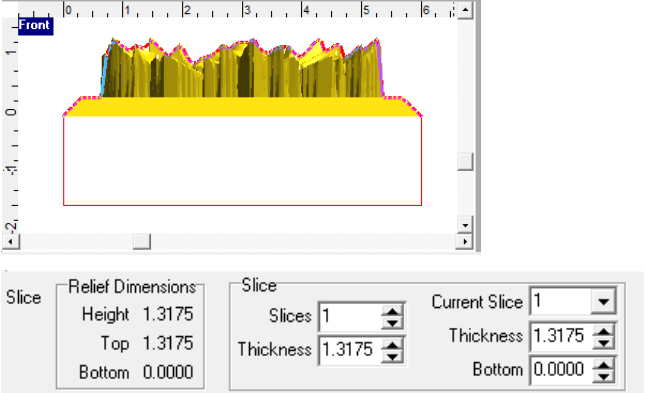
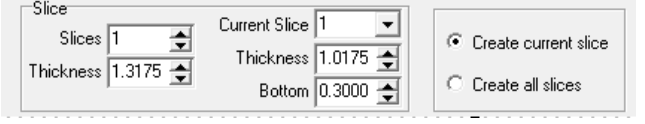
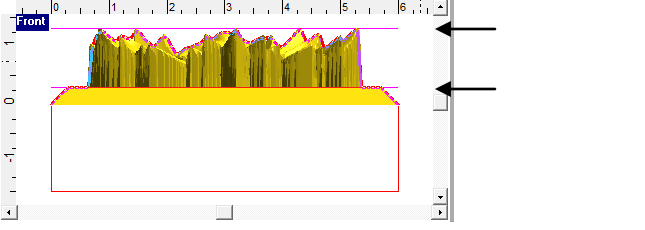

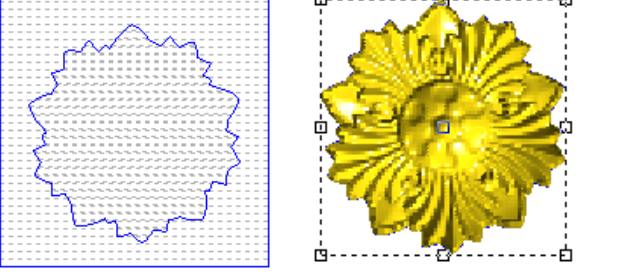
This example is showing how this design was extracted from the background relief.

1. Click and hold the Merge

 to open the flyout toolbar then click on the





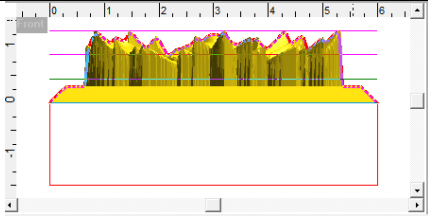
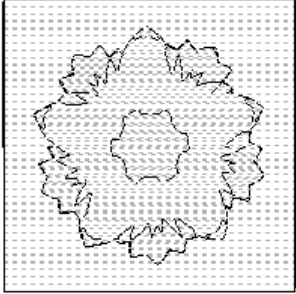
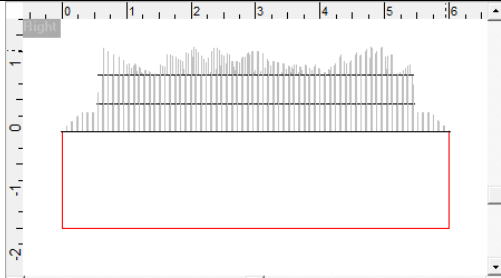
 This will open the Slices Dialog.



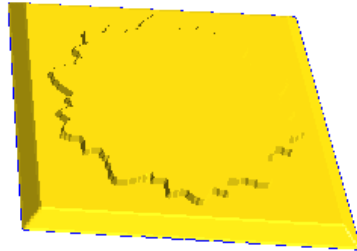
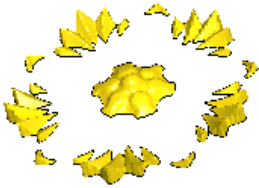
<p>2. Click on Front to work in the front view and then click on the relief. The relief information is loaded into the Slices Dialog.</p>	
<p>3. In this example, we created 1 slice with a thickness of 1.0175. The bottom of the slice is located .30 from the bottom of the relief.</p>	
<p>4. As you change the parameters you will notice that the pink lines represent the slice position.</p>	
<p>5. Click on the execute button.</p>	
<p>6. The first image of the top view shows the outline of the new contour that the slice has created.</p> <p>7. The second image shows the rendered view of the object.</p>	

Example 2:

This example is dividing the relief into 3.

<p>1. Click and hold the Merge Relief</p> <p>Icon  to open the flyout toolbar then click on the Slices</p> <p>Icon.  This will open the Slices Dialog.</p>	
<p>2. Change the number of slices to 3. This example shows the slices will each be .4392 thick. The Current Slice, slice 1 is shown with the pink lines. The bottom of the slice is positioned at .8783.</p>	
<p>3. You can see the layer lines in the front view. Change the Current slice number to get the exact location for each slice layer. The Create all slices parameters was selected, so each of the slice layers will be extracted.</p>	
<p>4. This is the top view of the slices that have been created.</p>	
<p>5. This is the front view of the slices that have been created. Notice the contour lines.</p>	

This image shows a rendered view of each of the separate slice layers that have been created.



18.Using Bitmaps



Introduction

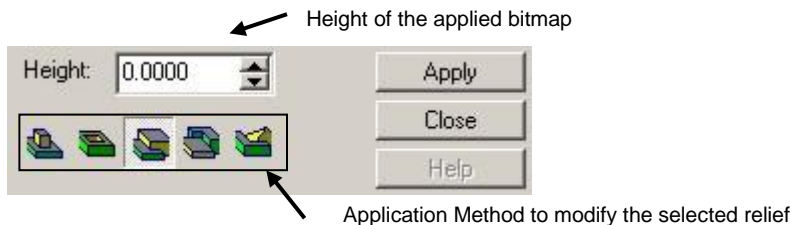
EnRoute utilizes bitmap images as a method for modifying relief surfaces. Bitmaps can be useful in creating a texture that makes up part of your design. They can also be useful as an efficient method for getting design information into EnRoute that might be difficult to create using other methods.



EnRoute utilizes the colors or grayscale shading in a bitmap image to modify the height of the relief using parameters that you provide. The process of applying a bitmap to a relief is simple, but it can be used in many different ways to create intricate surfaces.



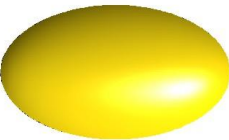

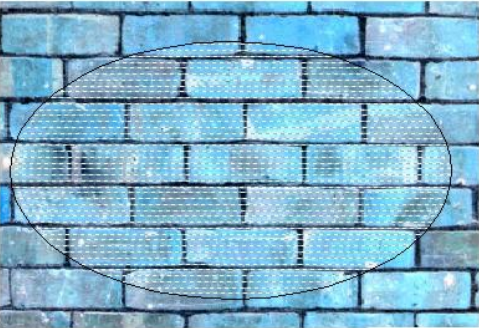







Applying Bitmaps to a Relief

The following graphic shows the dialog that is activated by the **Apply Bitmap**  tool. This function is activated by Selecting **Surface/Apply Bitmap**, or by selecting the **Apply Bitmap**  icon from the 3D Surfaces toolbar.



The **Application Method** defines how the bitmap will be applied to the relief surface. Typically you will select either the **Add**  or **Subtract**  method in order to apply either a positive or negative version of the bitmap. The **Height** parameter defines the maximum height of the bitmap as it is applied.

The following simple example illustrates the process of applying a bitmap to a relief.

<div>1. First create the relief that you want to add the bitmap to, and also import the bitmap file into your drawing. In this case, we have created an ellipse as the relief, and have imported a bitmap of a brick wall that will be used as a texture.</div>	<div>   </div>
<div>2. Position the bitmap on top of the relief.</div>	<div>  </div>
<div>3. With both the relief and the bitmap selected, choose the Apply Bitmap tool by selecting Surface/Apply bitmap .</div>	
<div>4. Choose the Application Method and the Height. In this case, choose Add, and define a Height of 0.20.</div>	<div> <div>Height: <input type="text" value="0.2000"/></div> <div>      </div> <div> <div>Apply</div> <div>Close</div> <div>Help</div> </div> </div>
<div>5. Click Apply.</div>	
<div>6. This image shows a rendered perspective view of the results.</div>	<div>  </div>

Using a Mask

When you apply a bitmap to a relief, you may not always want to apply the bitmap to the entire relief. EnRoute provides a method for masking the bitmap so that it applies only to the desired portion of the relief.

Any contour that is not part of the relief boundary or that does not have toolpaths applied to it can be used as a mask. It is only necessary to select the masking contours along with the relief and the bitmap when you activate the **Apply Bitmap** function. You will notice that the masking contours are automatically sorted into groups of containers and holes, and when the bitmap is applied only the portion of the bitmap that is inside the mask will be applied.

The following image shows a relief that had a bitmap texture applied using letters as a mask.

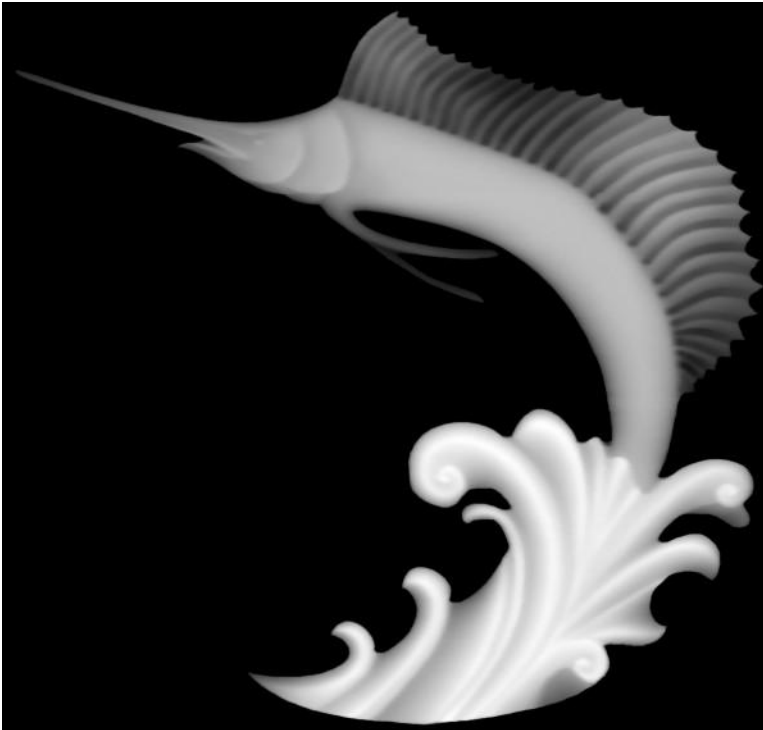


3D Effects using Bitmaps

It is possible to create realistic 3D effects using bitmaps, but this requires bitmap images that are created with this in mind. Remember that when a bitmap is applied, the colors and shading in the image are what control how the bitmap modifies the relief.

In a grayscale image, the different shades of gray on the scale from white to black define the height of the bitmap. The **Height** parameter that you define provides the overall height of the bitmap. With a grayscale image, white colors would be applied at this Height value, and then the shades of gray would be shorter down to black which has no effect on the relief.

The following image shows an example of a grayscale drawing that was created in order to have the proper effect on a relief.



You can see that the lightest shades in the waves will be the tallest parts of the bitmap. Since the fish is a little darker, it will be less tall, and the background is black so it won't affect the relief.

The next image shows a relief after this bitmap has been applied. You can see that the shading provides a finished relief that is quite realistic.



The next thought you may have is that it would be nice to be able to apply a photograph to a relief to get a 3D effect. It certainly is quite feasible to use a photographic image in a relief. Remember though that EnRoute uses the shading in the image to determine height. Something that is taller in the photograph may or may not be a lighter shade than objects that are further back in the image. Since a photograph is flat, it does not provide 3D information to the software to determine realistic heights.

The result you get with a photograph may or may not be what is desired for the subjects in the photograph. The following images show a photograph, and the relief surface that was created using the image.

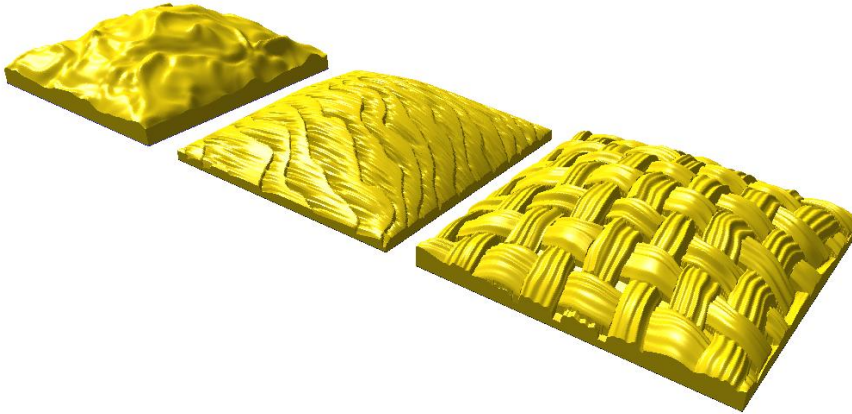


19.Texture Tools

Introduction

EnRoute also has the capability to create textures for a relief without the aid of bitmaps. While bitmaps can be useful in creating a texture that makes up part of your design they can be somewhat limited in that the size of the bitmap may not always work with your current design.

EnRoute has implemented a number of texture tools to design and create an unlimited number of possible textures to apply to a relief. Several designs have already been defined and entered in as templates. This is a good starting point for several different textures. Each of these textures can be used as defined or further refined to achieve the precise texture for your design. EnRoute then allows for the ability to save any new textures you have created as your own templates. This will allow you to build up an inventory of many different textures to retrieve for use with just a few clicks of the mouse.



Applying Textures to a Relief

Menu: Surface / Textures

Toolbar: 3D Surfaces / Textures

There are several types of textures available.

The categories include:

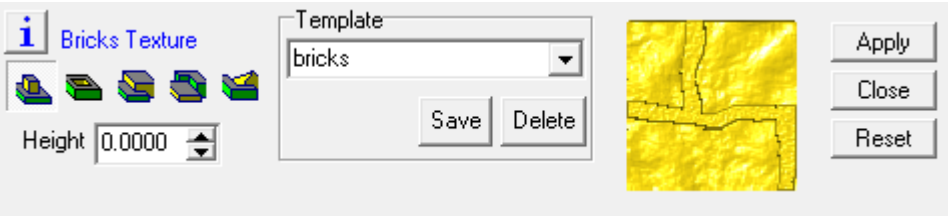
- Basic Noise
- Brick
- Cellular
- Dots
- FlagStone
- Flame
- Hammered
- Hexes




- Marble
- MudPot
- MultiCell
- MutiFract
- Terrain
- Veneer
- Weave
- Wood

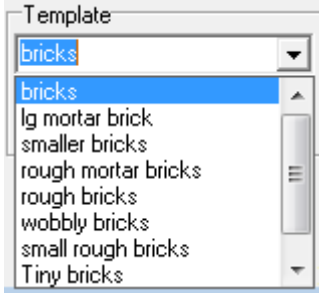
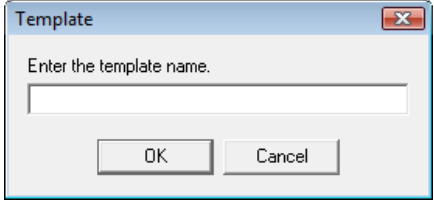
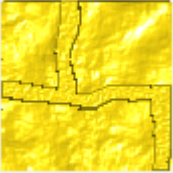
After selecting the relief, click to choose the type of texture to create. Each texture choice will bring up a dialog specific to that texture. Within the dialogs a template can be selected or specific parameters can be entered to create the desired texture.

Texture Dialog

The first section of the textures dialog will contain the same information for all of the texture types. The second section of the dialog will contain the parameters section that is specific to each type of texture. The dialog is used in the same manner for each of the different textures. The first section of the dialog is described below:



 Bricks Texture	This section of the dialog allows you to get specific information about the texture type that you have chosen. The name of the Texture is listed here and you can click on the information icon  to get specific information for each of the parameter settings.
	This section works the same way as the create relief section. Add, Subtract, Merge Highest, Merge Lowest, and Replace. See the create reliefs section for more information about these parameters.
Height	The height section specifies the height of the relief.
Template	The template section is used to create and store specific design parameters.

	<p>You can access some preloaded design templates by clicking the arrow for the drop down menu. Select any of these templates with the click of a mouse.</p> <p>The parameters will load for the template that you have chosen.</p>
<p>Save</p> 	<p>You can also create your own specific texture designs and name and save them to the template library.</p> <p>Once the new parameters have been entered;</p> <p>Click on the Save button.</p> <p>Enter the name of the template that you wish to save and click OK.</p>
<p>Delete</p>	<p>If you choose to delete a template, click on the name of the template that you want to delete and click the delete button. A dialog will open asking you to confirm that you want to delete this template.</p>
	<p>This section shows you a small preview of the parameters that you have entered. It is a nice tool to use while you are designing and creating new textures for your designs.</p>
<p>Apply</p>	<p>Press Apply to apply the texture to the selected relief.</p>
<p>Close</p>	<p>Click Close to close the Texture Dialog.</p>
<p>Reset</p>	<p>Reset will change the parameters back to the default parameters for the specific texture.</p>

Parameters Section of the Textures Dialog

Each of the Texture types will have its own set of parameters. The below example is the dialog for the bricks parameters.

Scale X: 1.0000 Y: 1.0000 Z: 1.0000	Position X: 0.0000 Y: 0.0000 Z: 0.0000	Brick Size X: 2.0000 Y: 1.0000 Z: 1.0000	Mortar Size X: 0.2000 Y: 0.2000 Z: 0.1000	Power: 1.0000 Jitter: 0.5000 Twist: 1.0000	Brick Rough: 0.1500 Mortar Rough: 0.0200
---	--	--	---	--	---

Click on the information icon to open the dialog for each specific texture. The Texture notes will give you information for each of the parameters specific to the texture you have selected.

Below is shown the Information Notes for the Bricks Texture. You can create your own templates for each of the textures using the guidelines here and the preview window.

EnRoute

Brick Texture Notes

This is a simple Bricks and Mortar texture that can be applied to reliefs. There are parameters to set the size and position of the bricks. Additional parameters allow you to set the roughness of the bricks and mortar as well as introducing some distortion into the brick pattern.

Parameters :

Position : Position of the texture.

Scale : The texture is scaled by this value. Use this one along with Brick Size and Mortar Size to get the effect that you want. The roughness and distortion of the bricks and mortar will also be scaled when you use this parameter.

Brick Size : Nominal dimensions for the bricks. Actual size of bricks is determined by the combination of Brick Size and Scale parameter.

Mortar Size : Nominal dimensions for the mortar. Actual size of mortar is determined by the combination of Mortar Size and Scale parameter.

Jitter : Strength of the distortion added to the brick pattern. Higher Jitter value will produce more distortion. Use this in combination with Twist parameter to define distortion.

Twist : This determines the tension in the distortion pattern. Higher values here make the distortion pattern smaller and 'tighter'. Use this in combination with the Jitter parameter to define distortion.

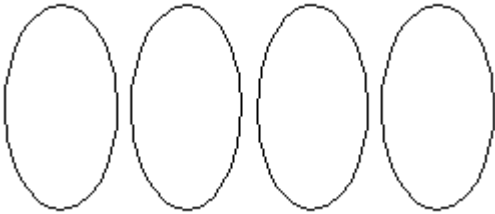
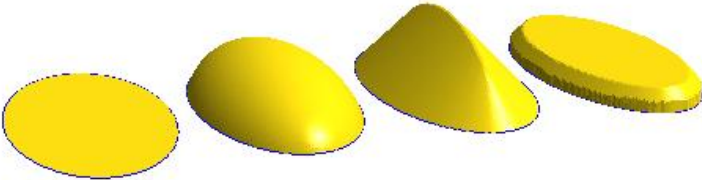




Brick Rough : Strength of the roughness applied to the bricks.

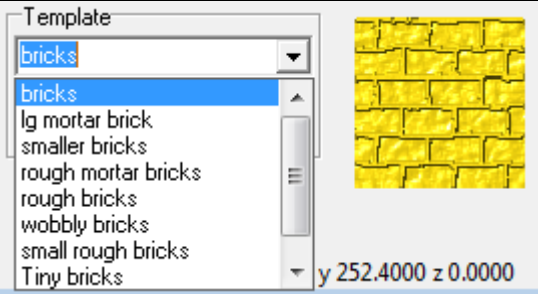



Mortar Rough : Strength of the roughness applied to the mortar.

Notes : If you get a combination of parameters that are acceptable and want to make the brick pattern bigger or smaller - use the Scale parameter to scale the bricks and distortion and roughness together. With the use of distortion, the brick texture can produce interesting patterns that look nothing like bricks and mortar.

OK

Example using the Brick Texture:

<div>1. In this example, the brick texture is shown on 4 different shapes of reliefs. Flat, Rounded, Beveled and Limited Height. Create simple a simple ellipse shape and use the Multicopy tool to copy the shape.</div>	
<div>2. Add the different relief shapes to the contour.</div>	
<div>3. Select the reliefs and click and hold the textures icon in the 3D Surfaces Toolbar. This will activate the flyout menu.</div> <div>4. Click on the Brick Texture Icon. This will open the Precision Toolbar.</div>	<div><div><div> EnRoute saves the last texture that was used to be the icon in the 3D Surfaces Toolbar. For example, if the last texture you used was the Brick Texture, then the brick icon would be the one in the 3D Surfaces toolbar and you would only need to click on the icon to activate the Brick Precision Toolbar. This helps to streamline the creative process.</div><div><div>Bricks Texture</div></div></div></div>
<div>5. Select the Parameters: Add to Relief.</div>	

<p>6. In this example we simply chose the brick texture from the template section.</p> <p>Click on the arrow to open the drop down list. As you highlight the choices in the list, the preview window will show you a small example of the texture.</p>	<div data-bbox="403 97 940 389">  </div> <p> You can also choose to create your own new texture by adjusting the parameters. The Information icon will give you specifics about each parameter and the preview window will allow you to see the changes as you develop the new texture.</p>
<p>7. Click on the Apply button to apply the texture to the reliefs.</p> <p>The textures will be applied to the reliefs and the Precision Toolbar will close.</p>	<div data-bbox="470 568 1102 747">  </div> <p> Notice how the texture is consistently applied to each of the different shapes.</p>



 Signs can be easily created using the textures with masks.

20.Relief Edit Tools


The New Relief Editing tools provide a quick and simple method for editing any relief surface using tools that are similar to bitmap painting tools. This provides another powerful creative tool for relief creation.

There are five new tools to use to edit your existing relief or to use to create a new relief using the mouse to draw a free hand design. The same basic concept of Add, Subtract, Merge Highest, Merge Lowest and Replace are used in combination with the shape of the tool specified.

Edit Relief Toolbar






Menu: Surface / Relief Edit

Toolbar: 3D Surfaces / Edit Selected Relief

These tools are found in the **Surface Menu** under **Relief Edit**, or in **3D Surfaces Toolbar** by selecting the **Edit Selected Relief** tool. 



Parameters:

	Smooth Relief Edit Tool – This tool is rounded at its tap and base so it works well to gently modify a relief surface.
	Bevel Relief Edit Tool – The bevel shape on this tool is great for adding details to your surface.
	Dome Relief Edit Tool – This tool has an arc shape.
	Flat Relief Edit Tool – This is good for interactively adding a specific amount to your surface.
	Blended Relief Edit Tool – This tool is used to blend the surface. It will even out the surface within the selected radius.
Radius	The radius specifies the size of the tool.
Angle	This is the angle from the edge of the tool to the center. This is used to define the height of the edit tool.
Height	This specifies the height of the flat tool.

Use Spline	If this option is checked, when you click and drag the mouse to edit the relief, EnRoute saves the points and then uses them to define a smooth curve for the edit. If it is unchecked then EnRoute uses the points it gets from the mouse directly without trying to smooth them.
Build Surface	If this parameter is checked, each time that you drag the tool over an area of the relief, the surface will be affected by the specific parameters of that tool and add to any prior movements of the mouse with that specific tool.
Fade	The fade parameter reduces the radius of the tool from the set size to zero within the stroke of the mouse movement. This allows you to blend with the existing relief.
Shrink	The shrink parameter affects the height of the tool parameters, reducing it from the highest to the lowest within the mouse movement. This also allows the tool to blend with the existing relief.
Apply	Click Apply to accept changes that have been made with an edit tool. These edits can be reversed outside of the edit tool by using undo.
Clear	Click the clear button if you choose not to accept the changes made while using the edit tools. If you change tools, the change is accepted for the previous tool. You can use the Undo tool if you do not like the changes to the relief that have been already been accepted.
Close	Click close to exit the tool.

Fade and Shrink

The fade and shrink parameters are used in conjunction with the Spline parameter. Using these parameters will help to blend into the existing relief. One thing to remember is that the larger the radius of the tool and the higher the resolution of the relief, the longer it will take to process.



This shows the bevel tool with the **Fade** parameter set all the way to the right. The mouse movement was from left to right. The tool edited the relief from the largest to the smallest point.



This shows the bevel tool with the **Shrink** parameter set all the way to the right. The mouse movement again was left to right. The relief does not get smaller in diameter, but the height of the relief moves from highest point to lowest point.

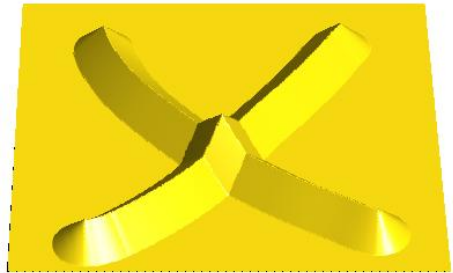


Build Parameter

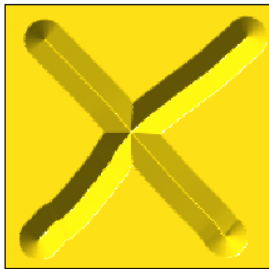
The build parameter allows you to use your tools to “build up” a specific area of your relief. In the examples below you can see how the relief is affected with and without the build parameter.



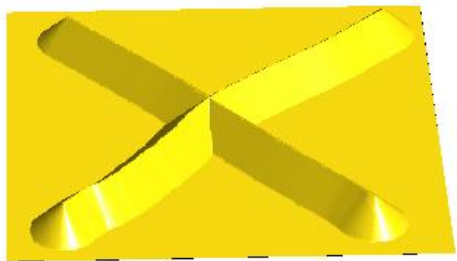
Top View with Build parameter checked.



Perspective View with Build parameter checked.



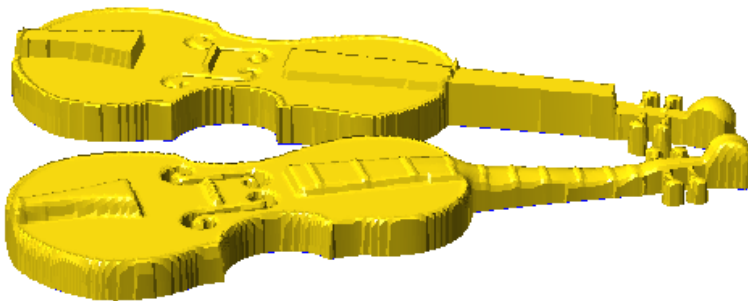
Top view with Build parameter unchecked.



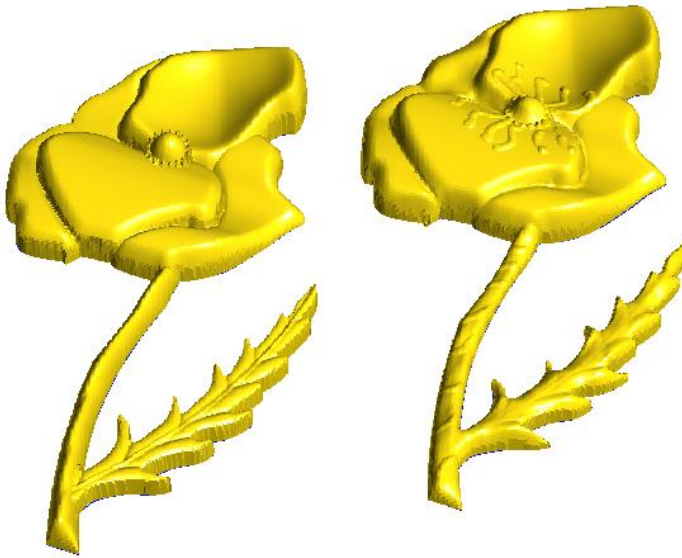
Perspective view with Build parameter unchecked.

Examples using relief edit tools

This example shows the use of the Relief Editing Tools. The top image is a quick assembly of a violin. Using the Relief Edit tools, you can add details and use the smoothing tool to blend and soften the design.



This image shows small adjustments to the relief.

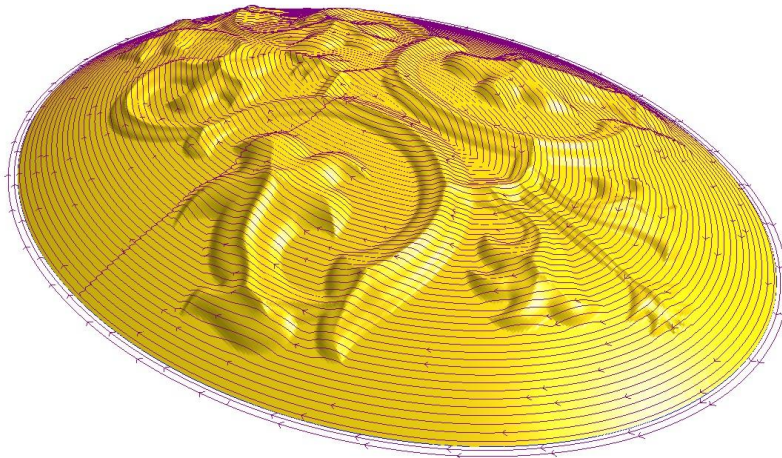


21. 3D Toolpaths

Introduction

After creating a relief, the next step is to create the toolpaths you will use to mill the finished piece. Although the results can be quite different, the process of creating 3D toolpaths is largely the same as creating other types of toolpaths in EnRoute. It includes selecting the proper strategy and then defining the tools and other parameters that will be used to create the toolpaths. The difference is that with 3D toolpaths there are a few additional parameters that are required to define how the relief is used to create the toolpaths.

This chapter describes how to use relief surfaces to create 3D toolpaths. It assumes the reader is already familiar with EnRoute 2D toolpath strategies.



3D Toolpath Strategies

3D toolpaths can be created using most of the same strategies that are used for other types of toolpaths. This includes routing offset, hatch fill, island fill, and engrave. The hatch fill and island fill strategies will be the ones you use most commonly since they are the ones that are used to mill the relief surfaces you create. Routing offsets are useful with reliefs to clean up tooling marks on vertical edges in the relief, and the engraving strategy allows you to add 3D engravings on relief surfaces.

3D Toolpath Strategy Parameter

The parameters that are required for defining 3D toolpaths are activated as part of the standard strategies if you have one or more reliefs selected when you activate that strategy.

Optimization	Standard	
Apply to relief	<input checked="" type="checkbox"/>	
Apply to surface	<input checked="" type="checkbox"/>	
Carve into surface	<input type="checkbox"/>	
Apply Overcut	<input checked="" type="checkbox"/>	

The first thing you notice when you activate a strategy with a relief selected is the new checkbox at the bottom of the parameters dialog, named **Apply to relief**. By checking this checkbox, you have access to four additional parameters that define how the toolpaths will be applied to the relief.

Apply to Surface

Apply to surface means that the toolpath will follow the surface of the relief and the depth of cut will be adjusted to follow the relief. If the toolpath covers an area with no relief, the depth is defined by the settings in the cut definition. The cut depth defines the maximum depth of the new toolpaths.

Carve into surface

Carve into surface allows the toolpath to follow the relief while using the cut depth information to define how deep to cut into the relief. One of the common uses of this parameter would be to create a 3D Engrave toolpath that follows the surface of the relief while creating the 3D effect into the relief surface.


When you select this option two more check boxes appear.


Project is the most common selection; it calculates the toolpaths as if they were applied/projected onto the relief.

Float calculates the toolpath and makes sure the tool will not remove part of the relief. This can cause distortion of the toolpaths and should only be used in special circumstances, if it is particularly important for the new toolpaths not to gouge into the relief.

Apply Overcut

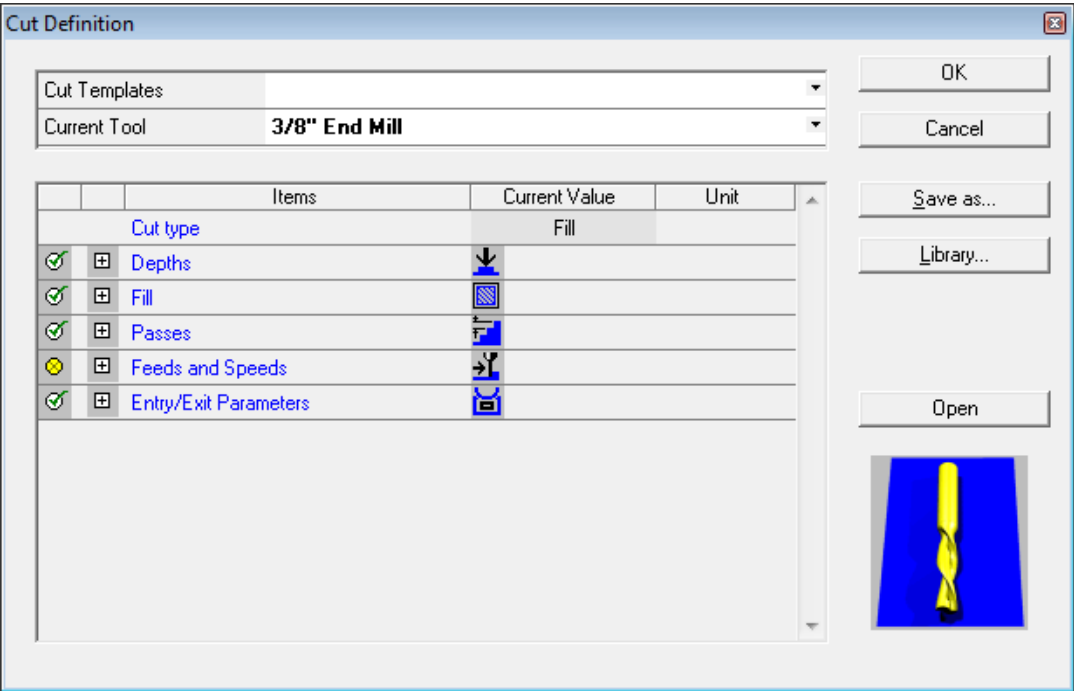
Overcut amount (Fills Only) allows EnRoute to compensate for the characteristics of the ball end mills used to create 3D objects. Without this parameter, the outside edges of your object would have a scalloped effect. Overcut allows the toolpath to be created *OUTSIDE* the boundaries of the relief to eliminate this problem.

 As a rule of thumb overcut should be set to a little less than ½ the tool diameter on the finish pass.

Apply to surface	<input checked="" type="checkbox"/>	
Carve into surface	<input type="checkbox"/>	
Apply Overcut	<input checked="" type="checkbox"/>	
Overcut amount	0.5000	

Cut Definition Parameters

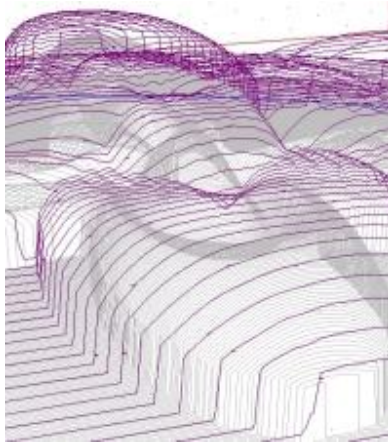
In the Cut Definition dialog there are a few more parameters that are new in 3D.



In the Depths section **Offset from surface** allows you to set how close you want the toolpaths to follow the surface of the relief. Usually this is defined for a roughing pass to ensure that the roughing pass does not damage the finished surface. The offset distance is determined by many factors, including type of material, type of tool (roughing and finish), and how fast you want the roughing pass to run.

Depths			
Surface		0.0000	in
Final Depth		0.7500	in
Offset from surface		0.2000	
Step Rough		<input type="checkbox"/>	

The following shows toolpaths with a 0.20 offset.

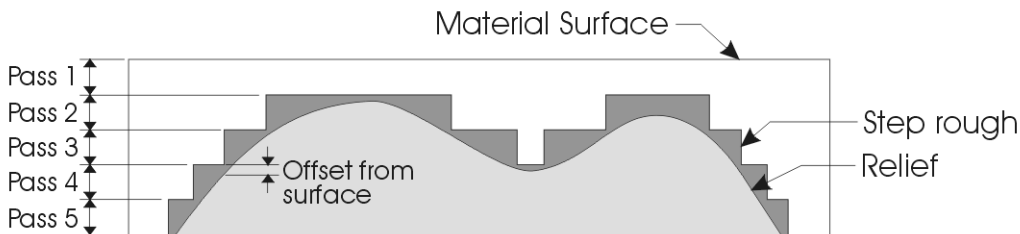


Step Rough

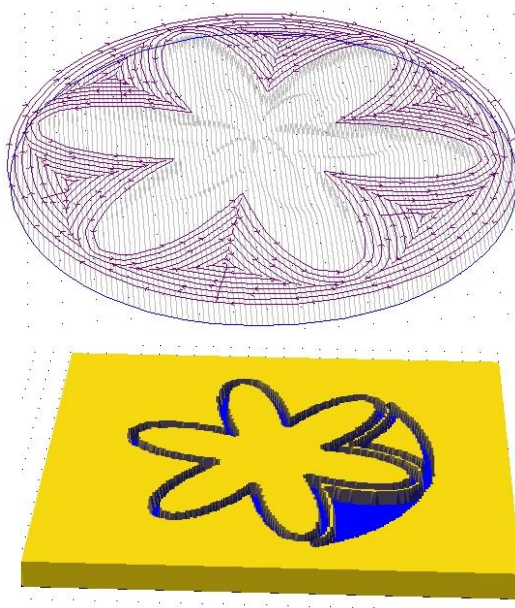
This option is available with Hatch and Island fills. It provides an optional method for creating toolpaths to rough out the relief surface prior to creating the finish toolpaths.

The advantage of this method is that it creates 2D toolpaths at multiple pass depths that remove as much material as possible at each pass depth. Because the toolpaths are 2D, they can typically be defined using higher feed rates than 3D toolpaths. Also, each pass only cuts the portions of the relief it can reach, so it is typically more efficient than using multiple passes of standard 3D fills as roughing passes.

The possible disadvantage of this method is that steps are left for the finish pass. The height of the steps is equal to the difference between depths of each pass of the step rough. The ball end mill that is used for the finish pass must be capable of cutting the steps that remain after the step rough.



The first image shows toolpaths with the step rough option. The second image shows a rendered view of these toolpaths.



Key 3D Toolpath Concepts

Before you start creating 3D toolpaths there are a few things to consider that mostly do not apply in 2D. You likely already understand that a three- axis router creates 3D objects with a ball end mill that “steps over” incrementally, making numerous passes at different depths to waste the material around the finished surface. The constraints here are the size of the tool, the amount of overlap, and how much detail is required. The relationship is the smaller the tool, the larger the overlap, the finer the detail, the LONGER the time to cut. EnRoute gives you the control you need to balance these factors so that you can create a high quality finished surface in a reasonable amount of time.

In general, the depths of cut are greater across greater areas of your material. This means you need to be careful to use the visualization tools in EnRoute to be sure your job will cut correctly. The rough tool and the roughing strategies will go a long way toward getting the job done. The reason for roughing is to clear material out of the way for the finish pass to run. You generally want to run the finish cut in a single pass as this pass takes the longest time to run; also, you have a more limited selection of tools for the finish cut. Another thing to consider is the length of the tool in relation to the surrounding material. It is easy to get yourself in trouble trying to get a small tool into a deep area. EnRoute gives you the tools to avoid these situations but you need to be aware that they exist.

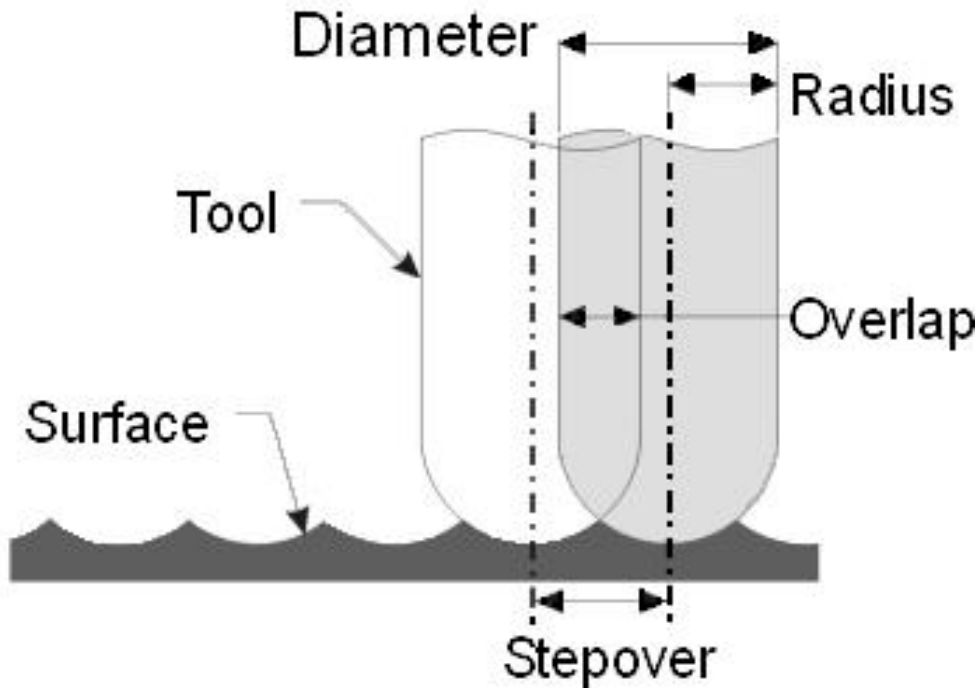
Overlap

When it comes time to mill out a 3D surface, the selection of the proper overlap percentage becomes quite important. The proper percentage can vary widely depending on the size of the relief, the size of the tool, the type of material being used, and the quality of the finished surface that is desired.

Milling a relief surface requires a ball end mill tool in order to allow you to cut smooth surfaces. The spherical surface of the tool will create small grooves in the finished surface. The size of the grooves will vary depending on the amount of overlap. As you increase overlap percentage the tool marks are reduced, but the time required to mill the surface increases. The goal is to choose an overlap value that balances cut quality with cut time.



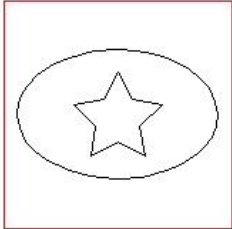



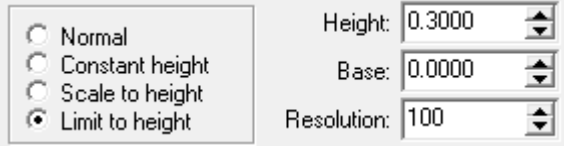

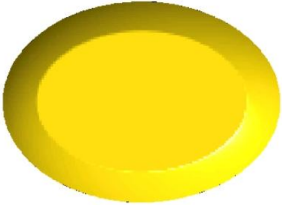

The best way to determine proper overlap for a job is to create and cut a test relief using the same type of material as the job. Then you can estimate the duration of the entire job, and see the surface quality, and the amount of finish work that will be necessary to complete the job.


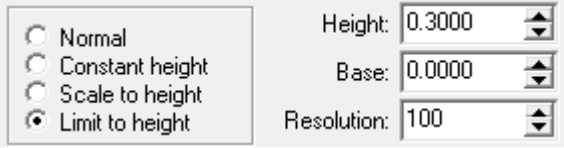



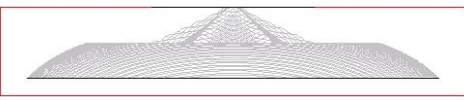
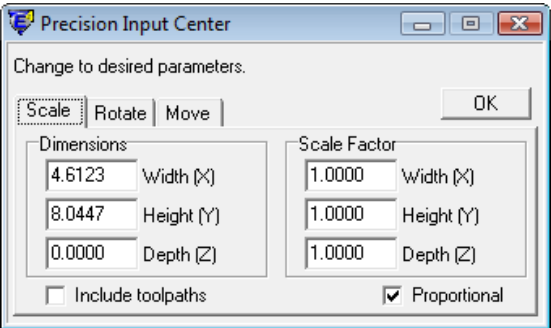
Typical overlap values range from 80% for larger reliefs that don't require a smooth finish, to 95% for small reliefs cut from dense material that requires a very smooth finish.


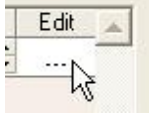


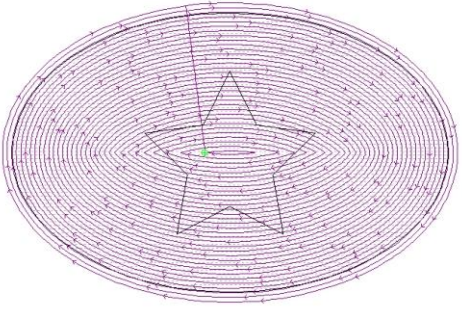
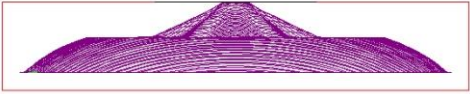
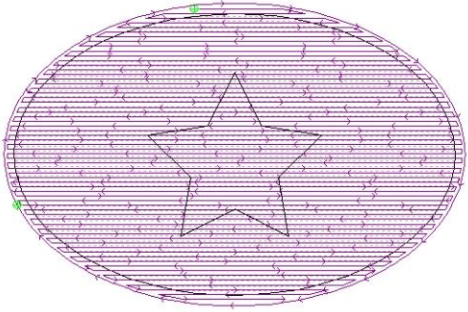
3D Toolpath Example

The following example illustrates the process of applying 3D toolpaths using the **Island Fill Strategy** and **Hatch Fill Strategy**.

<p>1. Start with a plate size of 4 x 4</p> <p>x 1.  Draw an oval and a star. Align the objects to center.</p> <p></p>	
<p>2. Select the Create a Relief icon.</p>	
<p>3. Click the Add icon to select that option.</p>	
<p>4. Select the Rounded option.</p>	
<p>5. Select the Limit to height Application method.</p> <p>Define Height = .30 Base =0.00 Resolution = 100</p>	
<p>6. Enter the angle parameter as 45.00. Click Apply.</p>	
<p>7. This is a rendered top view of the object.</p>	
<p>8. Select the relief, hold the Shift key and select the star. Select the Add Icon. By selecting both the relief and the star, this tells EnRoute to apply the star to the relief rather than creating a new separate relief for the star.</p>	

<p>9. Select the Beveled option.</p>	
<p>10. Select the Limit to Height application Method.</p> <p>Define: Height = .30 Base =0.00</p>	
<p>11. Enter the angle parameter as 45.00.</p>	
<p>12. The rendered view should look similar to this.</p>	
<p>13. Select the relief. Click and hold the Align relief icon. Select the Align all reliefs to top of plate option.</p>	
<p>14. This will move the relief into the plate vertically. The front view shows the vertical placement of the relief in the plate.</p>	
<p>15. You may also place the relief precisely by pressing F2 to activate the Precision Input Center.</p> <p>Click on the Move tab and enter the desired X-position of the relief.</p> <p>This option works best when the Front view is active, because then the button box selection applies to the X and Z location of the relief.</p>	

<p>16. Select Island Fill.  Add a ¼ inch ball nose tool.</p> <p>When a relief is selected, the option Apply to relief is activated in the parameters section. Make sure this is checked. Choose Apply to surface and Apply overcut. Enter overcut amount of 0.08.</p> <p>Usually the overcut amount should be set at a little less than one-half of the diameter of the tool.</p>	<div>Island Fill Parameters:</div> <table><tr><td>Optimization</td><td>Standard</td><td>▼</td></tr><tr><td>Apply to relief</td><td><input checked="" type="checkbox"/></td><td></td></tr><tr><td>Apply to surface</td><td><input checked="" type="checkbox"/></td><td></td></tr><tr><td>Carve into surface</td><td><input type="checkbox"/></td><td></td></tr><tr><td>Apply Overcut</td><td><input checked="" type="checkbox"/></td><td></td></tr><tr><td>Overcut amount</td><td>0.08</td><td>▲▼</td></tr></table>	Optimization	Standard	▼	Apply to relief	<input checked="" type="checkbox"/>		Apply to surface	<input checked="" type="checkbox"/>		Carve into surface	<input type="checkbox"/>		Apply Overcut	<input checked="" type="checkbox"/>		Overcut amount	0.08	▲▼
Optimization	Standard	▼																	
Apply to relief	<input checked="" type="checkbox"/>																		
Apply to surface	<input checked="" type="checkbox"/>																		
Carve into surface	<input type="checkbox"/>																		
Apply Overcut	<input checked="" type="checkbox"/>																		
Overcut amount	0.08	▲▼																	
<p>17. Click in the edit box for the ¼ inch tool to activate the Cut Definitions dialog.</p>																			
<p>18. Enter a final depth of 0.60.</p>	<div><div>Depths</div><div><table><tr><td>Surface</td><td>0.0000</td><td>▲▼</td></tr><tr><td>Final Depth</td><td>0.0600</td><td>▲▼</td></tr><tr><td>Offset from surface</td><td>0.0000</td><td>▲▼</td></tr><tr><td>Step Rough</td><td><input type="checkbox"/></td><td></td></tr></table></div></div>	Surface	0.0000	▲▼	Final Depth	0.0600	▲▼	Offset from surface	0.0000	▲▼	Step Rough	<input type="checkbox"/>							
Surface	0.0000	▲▼																	
Final Depth	0.0600	▲▼																	
Offset from surface	0.0000	▲▼																	
Step Rough	<input type="checkbox"/>																		
<p>19. Enter fill overlap of 85%. For a finished surface, the overlap percentage will typically vary between 80% and 95%, depending on the material, and the quality of the finished surface that is required.</p>	<div><div>Fill</div><div><table><tr><td>Overlap</td><td>85.0000</td><td>▲▼ %</td></tr><tr><td>Corner tags</td><td><input type="checkbox"/></td><td></td></tr></table></div></div>	Overlap	85.0000	▲▼ %	Corner tags	<input type="checkbox"/>													
Overlap	85.0000	▲▼ %																	
Corner tags	<input type="checkbox"/>																		
<p>20. Enter number of passes as 1. In this example we are not defining a rough cut tool, and the assumption is that the ¼ inch ball end mill tool will be capable of cutting the full depth of the relief in one pass.</p>	<div><div>Passes</div><div><table><tr><td>Number</td><td>1</td><td>▲▼</td></tr></table></div></div>	Number	1	▲▼															
Number	1	▲▼																	

<p>21. Click Ok to return to the Island Fill Dialog. Click OK to calculate the tool paths.</p> <p>This is the top view of the Island Fill tool paths.</p>	
<p>22. This is the front view.</p> <p>To cut the object from the material, a Routing Offset toolpath would be defined.</p>	
<p>23. This example shows the same relief with Hatch Fill toolpaths applied. The same parameters have been applied when defining the toolpaths.</p> <p>The preferred strategy would be determined by the type of material used. The cutting time in this instance would likely be shorter with the Island Fill strategy.</p>	






3D Toolpaths with a Step Rough

This next example demonstrates the application of toolpaths to a larger and more intricate design.






This type of relief requires a little more thought when creating toolpaths. Because it is deep, it likely isn't possible to cut it out using one tool. Therefore, it is necessary to create one or two roughing passes to remove the bulk of the waste material prior to running the finish toolpath.

The first cut we add is a rough pass. The primary purpose of this cut is to remove material efficiently. The step rough option allows for the most efficient removal of material.

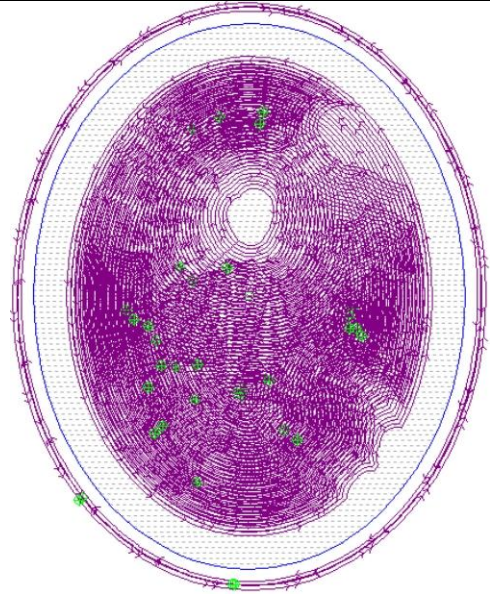
The following relief will be cut using the **Island Fill** strategy. The first pass utilizes the step rough feature. This allows for most of the material to be cleared away prior to the finishing passes. The parameters are defined as follows.

<div>1. This is the top view of the relief that will be toolpathed in this example.</div> <div>The size of the relief is approximately 12 x 15 x 2 inches.</div>									
<div>2. This is the Front view of the relief in this example.</div>									
<div>3. A 3/8 inch end mill tool is used for the step rough toolpaths.</div>	<table><tr><th></th><th>Order</th><th>Tool</th><th>T</th></tr><tr><td></td><td>1</td><td>3/8" End Mill</td><td></td></tr></table>		Order	Tool	T		1	3/8" End Mill	
	Order	Tool	T						
	1	3/8" End Mill							

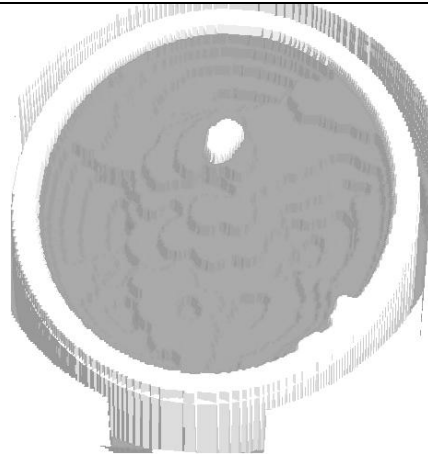
The following parameters were entered to create the Island Fill toolpaths.

<p>4. Select the following: Apply to Relief Apply to Surface Apply Overcut Overcut amount = 0.50</p> <p>5.</p>	<p>Island Fill Parameters:</p> <table border="1"> <tr> <td>Optimization</td> <td>Standard</td> <td>▼</td> </tr> <tr> <td>Apply to relief</td> <td><input checked="" type="checkbox"/></td> <td></td> </tr> <tr> <td>Apply to surface</td> <td><input checked="" type="checkbox"/></td> <td></td> </tr> <tr> <td>Carve into surface</td> <td><input type="checkbox"/></td> <td></td> </tr> <tr> <td>Apply Overcut</td> <td><input checked="" type="checkbox"/></td> <td></td> </tr> <tr> <td>Overcut amount</td> <td>0.5000</td> <td>⬆ ⬇ ⬆</td> </tr> </table>	Optimization	Standard	▼	Apply to relief	<input checked="" type="checkbox"/>		Apply to surface	<input checked="" type="checkbox"/>		Carve into surface	<input type="checkbox"/>		Apply Overcut	<input checked="" type="checkbox"/>		Overcut amount	0.5000	⬆ ⬇ ⬆
Optimization	Standard	▼																	
Apply to relief	<input checked="" type="checkbox"/>																		
Apply to surface	<input checked="" type="checkbox"/>																		
Carve into surface	<input type="checkbox"/>																		
Apply Overcut	<input checked="" type="checkbox"/>																		
Overcut amount	0.5000	⬆ ⬇ ⬆																	
<p>6. Define the Depths as: Surface = 0.00 Final Depth = 2.0 Offset from surface = 0.20 Step Rough = checked</p>	<p>Depths </p> <table border="1"> <tr> <td>Surface</td> <td>0.0000</td> </tr> <tr> <td>Final Depth</td> <td>2.0000</td> </tr> <tr> <td>Offset from surface</td> <td>0.0200</td> </tr> <tr> <td>Step Rough</td> <td><input checked="" type="checkbox"/></td> </tr> </table>	Surface	0.0000	Final Depth	2.0000	Offset from surface	0.0200	Step Rough	<input checked="" type="checkbox"/>										
Surface	0.0000																		
Final Depth	2.0000																		
Offset from surface	0.0200																		
Step Rough	<input checked="" type="checkbox"/>																		
<p>7. Overlap of 75%. Corner tags = unchecked</p>	<p>Fill </p> <table border="1"> <tr> <td>Overlap</td> <td>75.0000</td> </tr> <tr> <td>Corner tags</td> <td><input type="checkbox"/></td> </tr> </table>	Overlap	75.0000	Corner tags	<input type="checkbox"/>														
Overlap	75.0000																		
Corner tags	<input type="checkbox"/>																		
<p>8. Passes are defined as: Number = 8 Maximum per Pass = 3.0 Actual per Pass = 0.25 Final Pass = unchecked Final Pass Depth = 0.00</p>	<p>Passes </p> <table border="1"> <tr> <td>Number</td> <td>8</td> </tr> <tr> <td>Maximum per Pass</td> <td>3.0000</td> </tr> <tr> <td>Actual per Pass</td> <td>0.2500</td> </tr> <tr> <td>Final Pass</td> <td><input type="checkbox"/></td> </tr> <tr> <td>Final Pass Depth</td> <td>0.0000</td> </tr> </table>	Number	8	Maximum per Pass	3.0000	Actual per Pass	0.2500	Final Pass	<input type="checkbox"/>	Final Pass Depth	0.0000								
Number	8																		
Maximum per Pass	3.0000																		
Actual per Pass	0.2500																		
Final Pass	<input type="checkbox"/>																		
Final Pass Depth	0.0000																		
<p>9. The following feed rates are defined. This part is to be cut from Western Red Cedar, which is a soft wood that is prone to chipping.</p> <p>Feed Rate = 220.00 Final Pass Feed = 0.00 Plunge Rate = 100.0 Dwell = 0.00 Spindle = 18000</p>	<p>Feeds and Speeds </p> <table border="1"> <tr> <td>Feed Rate</td> <td>220.0000</td> </tr> <tr> <td>Final Pass Feed</td> <td>0.0000</td> </tr> <tr> <td>Plunge Rate</td> <td>100.0000</td> </tr> <tr> <td>Dwell</td> <td>0.0000</td> </tr> <tr> <td>Spindle</td> <td>18000</td> </tr> </table>	Feed Rate	220.0000	Final Pass Feed	0.0000	Plunge Rate	100.0000	Dwell	0.0000	Spindle	18000								
Feed Rate	220.0000																		
Final Pass Feed	0.0000																		
Plunge Rate	100.0000																		
Dwell	0.0000																		
Spindle	18000																		
<p>10. Select the Climb direction.</p>	<p>Direction </p> <table border="1"> <tr> <td>Conventional</td> <td><input type="checkbox"/></td> </tr> <tr> <td>Climb</td> <td><input checked="" type="checkbox"/></td> </tr> </table>	Conventional	<input type="checkbox"/>	Climb	<input checked="" type="checkbox"/>														
Conventional	<input type="checkbox"/>																		
Climb	<input checked="" type="checkbox"/>																		

11. The toolpaths for the 3/8 inch tool are shown here.






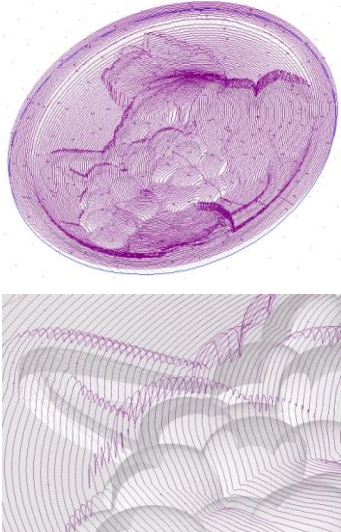


12. This image shows a rendered view of the step rough toolpath using the 3/8-inch tool.



Roughing Pass

The next step in defining the toolpaths for this relief is another roughing pass. The second cut to add is another rough cut. The purpose here is to create a surface that is closer to the finished surface than the step rough. One disadvantage of a step rough is that it creates a surface with ledges that may be too tall for the finish tool we will use. A standard roughing pass will create a relatively smooth surface.

13. Select the Island Fill Strategy and a ¼ inch end mill tool for this cut.																			
14. Select the following: Apply to Relief Apply to Surface Apply Overcut Overcut amount = 0.50	Island Fill Parameters: <table><tr><td>Optimization</td><td>Standard</td><td>▼</td></tr><tr><td>Apply to relief</td><td><input checked="" type="checkbox"/></td><td></td></tr><tr><td>Apply to surface</td><td><input checked="" type="checkbox"/></td><td></td></tr><tr><td>Carve into surface</td><td><input type="checkbox"/></td><td></td></tr><tr><td>Apply Overcut</td><td><input checked="" type="checkbox"/></td><td></td></tr><tr><td>Overcut amount</td><td>0.5000</td><td>⬆ ⬇ ⬆</td></tr></table>	Optimization	Standard	▼	Apply to relief	<input checked="" type="checkbox"/>		Apply to surface	<input checked="" type="checkbox"/>		Carve into surface	<input type="checkbox"/>		Apply Overcut	<input checked="" type="checkbox"/>		Overcut amount	0.5000	⬆ ⬇ ⬆
Optimization	Standard	▼																	
Apply to relief	<input checked="" type="checkbox"/>																		
Apply to surface	<input checked="" type="checkbox"/>																		
Carve into surface	<input type="checkbox"/>																		
Apply Overcut	<input checked="" type="checkbox"/>																		
Overcut amount	0.5000	⬆ ⬇ ⬆																	
15. Surface = 0.00 Final depth = 2.00 Offset from surface = .10 Step Rough = unchecked	Depths  <table><tr><td>Surface</td><td>0.0000</td></tr><tr><td>Final Depth</td><td>2.0000</td></tr><tr><td>Offset from surface</td><td>0.1000</td></tr><tr><td>Step Rough</td><td><input type="checkbox"/></td></tr></table>	Surface	0.0000	Final Depth	2.0000	Offset from surface	0.1000	Step Rough	<input type="checkbox"/>										
Surface	0.0000																		
Final Depth	2.0000																		
Offset from surface	0.1000																		
Step Rough	<input type="checkbox"/>																		
16. Overlap = 75% Corner tags = checked An overlap of 75% gives a relatively smooth surface without requiring too much time to cut.	Fill  <table><tr><td>Overlap</td><td>75.0000</td></tr><tr><td>Corner tags</td><td><input type="checkbox"/></td></tr></table>	Overlap	75.0000	Corner tags	<input type="checkbox"/>														
Overlap	75.0000																		
Corner tags	<input type="checkbox"/>																		
17. Passes, Number = 1 Maximum per Pass = 2.1 Actual per Pass = 2.0 Final pass = unchecked Final Pass Depth = 0.0	Passes  <table><tr><td>Number</td><td>1</td></tr><tr><td>Maximum per Pass</td><td>2.1000</td></tr><tr><td>Actual per Pass</td><td>2.0000</td></tr><tr><td>Final Pass</td><td><input type="checkbox"/></td></tr><tr><td>Final Pass Depth</td><td>0.0000</td></tr></table>	Number	1	Maximum per Pass	2.1000	Actual per Pass	2.0000	Final Pass	<input type="checkbox"/>	Final Pass Depth	0.0000								
Number	1																		
Maximum per Pass	2.1000																		
Actual per Pass	2.0000																		
Final Pass	<input type="checkbox"/>																		
Final Pass Depth	0.0000																		

<p>18. Feed Rate = 280.0 Final Pass Feed = 0.00 Plunge Rate = 100.0 Dwell = 0.00 Spindle = 18000</p>	<table border="1"> <thead> <tr> <th colspan="2">Feeds and Speeds</th> </tr> </thead> <tbody> <tr> <td>Feed Rate</td><td>280.0000</td></tr> <tr> <td>Final Pass Feed</td><td>0.0000</td></tr> <tr> <td>Plunge Rate</td><td>100.0000</td></tr> <tr> <td>Dwell</td><td>0.0000</td></tr> <tr> <td>Spindle</td><td>18000</td></tr> </tbody> </table>	Feeds and Speeds		Feed Rate	280.0000	Final Pass Feed	0.0000	Plunge Rate	100.0000	Dwell	0.0000	Spindle	18000
Feeds and Speeds													
Feed Rate	280.0000												
Final Pass Feed	0.0000												
Plunge Rate	100.0000												
Dwell	0.0000												
Spindle	18000												
<p>19. Select the Climb direction.</p>	<table border="1"> <thead> <tr> <th colspan="2">Direction</th> </tr> </thead> <tbody> <tr> <td>Conventional</td><td><input type="checkbox"/></td></tr> <tr> <td>Climb</td><td><input checked="" type="checkbox"/></td></tr> </tbody> </table>	Direction		Conventional	<input type="checkbox"/>	Climb	<input checked="" type="checkbox"/>						
Direction													
Conventional	<input type="checkbox"/>												
Climb	<input checked="" type="checkbox"/>												
<p>20. In the perspective view; you can see that we have created one pass that follows the surface. Notice that the toolpaths are not directly ON the surface. This is because we used an end mill and the program calculates to the edge of the tool but the toolpaths are displayed as centerlines. Normally the roughing pass would be set up with an offset, which creates a toolpath that keeps the edge of the tool away from the relief by the offset amount. In this example we used an offset amount of 0.10 inch.</p>													
<p>21. This is the front view of the roughing pass toolpaths.</p>													
<p>22. This is a rendered view of the step rough pass and the roughing pass toolpaths.</p>													

Finish Pass

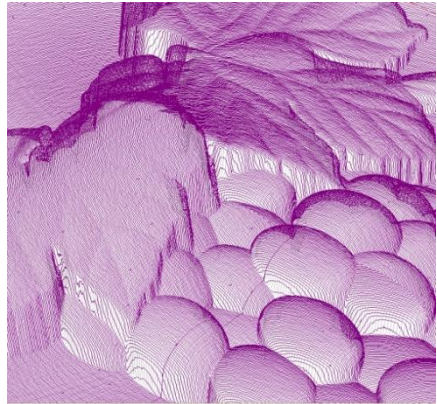
The final step in cutting this relief is to add the finish pass. For this pass a 1/8 inch ball end mill tool has been defined. You can create one strategy that includes both the roughing and finish passes, but you may prefer to create them separately since this can allow easier visualization and control later.

23. Select the following: Apply to relief Apply to surface Apply overcut Overcut = 0.10											
24. Surface = 0.00 Final depth = 2.0 Offset from surface = 0.00 Step Rough = unchecked Notice with the finish pass that the offset is set to 0.00.	<div>Depths</div> <table><tr><td>Surface</td><td>0.0000</td></tr><tr><td>Final Depth</td><td>2.0000</td></tr><tr><td>Offset from surface</td><td>0.0000</td></tr><tr><td>Step Rough</td><td><input type="checkbox"/></td></tr></table>	Surface	0.0000	Final Depth	2.0000	Offset from surface	0.0000	Step Rough	<input type="checkbox"/>		
Surface	0.0000										
Final Depth	2.0000										
Offset from surface	0.0000										
Step Rough	<input type="checkbox"/>										
25. Overlap = 88% Corner tags = unchecked	<table><tr><td>Overlap</td><td>88.0000</td></tr><tr><td>Hatch angle</td><td>0.0000</td></tr></table>	Overlap	88.0000	Hatch angle	0.0000						
Overlap	88.0000										
Hatch angle	0.0000										
26. Passes: Number = 1 Maximum per Pass = 2.10 Actual per Pass = 2.0 Final pass = unchecked Final Pass Depth 0.00	<div>Passes</div> <table><tr><td>Number</td><td>1</td></tr><tr><td>Maximum per Pass</td><td>2.1000</td></tr><tr><td>Actual per Pass</td><td>2.0000</td></tr><tr><td>Final Pass</td><td><input type="checkbox"/></td></tr></table>	Number	1	Maximum per Pass	2.1000	Actual per Pass	2.0000	Final Pass	<input type="checkbox"/>		
Number	1										
Maximum per Pass	2.1000										
Actual per Pass	2.0000										
Final Pass	<input type="checkbox"/>										
27. Feed Rate = 360.00 Final Pass Feed = 0.00 Plunge Rate = 100.0 Dwell = 0.00 Spindle = 18000	<div><div>Feeds and Speeds</div><table><tr><td>Feed Rate</td><td>360.0000</td></tr><tr><td>Final Pass Feed</td><td>0.0000</td></tr><tr><td>Plunge Rate</td><td>100.0000</td></tr><tr><td>Dwell</td><td>0.0000</td></tr><tr><td>Spindle</td><td>18000</td></tr></table></div>	Feed Rate	360.0000	Final Pass Feed	0.0000	Plunge Rate	100.0000	Dwell	0.0000	Spindle	18000
Feed Rate	360.0000										
Final Pass Feed	0.0000										
Plunge Rate	100.0000										
Dwell	0.0000										
Spindle	18000										
28. Select the climb direction.	<div>Direction</div> <table><tr><td>Conventional</td><td><input type="checkbox"/></td></tr><tr><td>Climb</td><td><input checked="" type="checkbox"/></td></tr></table>	Conventional	<input type="checkbox"/>	Climb	<input checked="" type="checkbox"/>						
Conventional	<input type="checkbox"/>										
Climb	<input checked="" type="checkbox"/>										

29. This is a rendered view including all toolpaths.




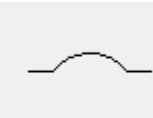




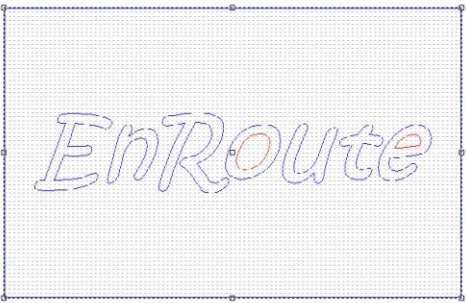



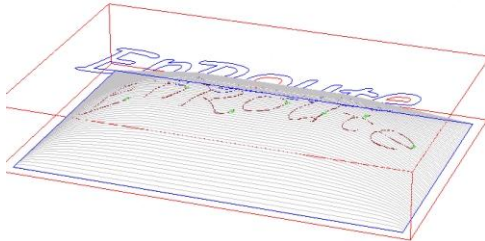


30. This is a close-up view of the 3D toolpaths. Rotate it in perspective and you will see the dense pattern of your finish pass toolpaths. The appropriate overlap amount varies depending on the material you are cutting, the amount of cleanup work you want to do, and the length of time you have to run the job. Values of 75% (coarse) to 95% (fine) are typical. The higher the overlap the more time the job will require to run. We suggest that you experiment by running the same relief at different overlaps to get a feel for the time and texture.





Engrave Toolpaths on a Relief

This next example shows the application of 3D toolpaths applied directly to a relief.

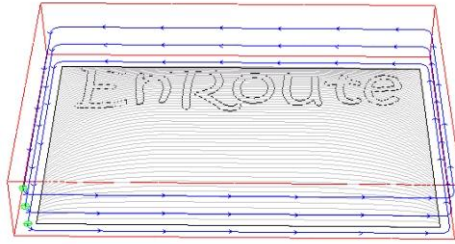
1. In this example the plate size is 4 x 6 x 1. The object is a 4 x 6 inch rectangle.	
2. Select the rectangle. Create a relief  Add to relief Rounded relief Normal Resolution = 100 Angle = 50 degrees	  <div><div><input checked="" type="radio"/> Normal <input type="radio"/> Constant height <input type="radio"/> Scale to height <input type="radio"/> Limit to height</div><div>Height: 0.0000 Base: 0.0000 Resolution: 100</div></div> <div><div>Angle: 50.0000</div></div>
3. The results should be similar to this perspective view image.	
4. Move the relief into the plate. Select the relief. Click and hold the Align relief icon. Select the Align all reliefs to top of plate option.	
5.  Add a text object to the design.  Center the text object with the relief.	

<div>6. Select both the relief and the text object.</div> <div>Select the Engrave toolpath icon. </div> <div>Add a Conic Tool with a Depth of .10.</div>	<table><tr><th>Tool</th><th>Type</th><th>Depth</th></tr><tr><td>90 Deg Conic</td><td>Engrave</td><td>0.1000</td></tr></table>	Tool	Type	Depth	90 Deg Conic	Engrave	0.1000												
Tool	Type	Depth																	
90 Deg Conic	Engrave	0.1000																	
<div>7. Select the following:</div> <div>3D Engrave toolpath Internal Apply to relief Carve into surface. The parameter here is the choice to carve into the surface rather than to apply to the surface.</div>	<div>Engrave Parameters:</div> <table><tr><td>3D engrave toolpath</td><td><input checked="" type="checkbox"/></td></tr><tr><td>Follow contour</td><td><input type="checkbox"/></td></tr><tr><td>External (male)</td><td><input type="checkbox"/></td></tr><tr><td>Internal (female)</td><td><input checked="" type="checkbox"/></td></tr><tr><td>Apply relief</td><td><input checked="" type="checkbox"/></td></tr><tr><td>Apply to surface</td><td><input type="checkbox"/></td></tr></table> <table><tr><td>Carve into surface</td><td><input checked="" type="checkbox"/></td></tr><tr><td>Project</td><td><input checked="" type="checkbox"/></td></tr><tr><td>Float</td><td><input type="checkbox"/></td></tr></table>	3D engrave toolpath	<input checked="" type="checkbox"/>	Follow contour	<input type="checkbox"/>	External (male)	<input type="checkbox"/>	Internal (female)	<input checked="" type="checkbox"/>	Apply relief	<input checked="" type="checkbox"/>	Apply to surface	<input type="checkbox"/>	Carve into surface	<input checked="" type="checkbox"/>	Project	<input checked="" type="checkbox"/>	Float	<input type="checkbox"/>
3D engrave toolpath	<input checked="" type="checkbox"/>																		
Follow contour	<input type="checkbox"/>																		
External (male)	<input type="checkbox"/>																		
Internal (female)	<input checked="" type="checkbox"/>																		
Apply relief	<input checked="" type="checkbox"/>																		
Apply to surface	<input type="checkbox"/>																		
Carve into surface	<input checked="" type="checkbox"/>																		
Project	<input checked="" type="checkbox"/>																		
Float	<input type="checkbox"/>																		
<div>8. After entering the appropriate strategy and cut parameters, click OK in the strategy dialog to create the toolpaths.</div>																			
<div>9. Look at the job in the Perspective view and see how the contours stay at the top of the plate while the toolpaths follow the underlying relief. This tool allows us to create a 2 1/2D toolpath along a 3D object. Remember, a key to this technique is to select the relief as well as the object when you first activate the toolpath dialog.</div>																			
<div>10.  This is a rendered view of the object after the relief surface has been toolpathed using a hatch fill and the engrave toolpaths have been applied.</div> <div>Notice how the engraved word follows the curve of the relief.</div>																			

Routing Offset

<div>1. Use the same relief as in the previous example to create a routing offset toolpath. In this example, you will see that creating a routing offset to cut out a relief is no different than creating any other type of routing offset.</div>																			
<div>2. Select the relief. Select the Routing Offset icon.</div> <div> Add a 1/4 inch End Mill tool, with a depth of 1.00 .</div>	<table><tr><th>Tool</th><th>Type</th><th>Depth</th></tr><tr><td>1/4" End Mill</td><td>Rough</td><td>1.0000</td></tr></table>	Tool	Type	Depth	1/4" End Mill	Rough	1.0000												
Tool	Type	Depth																	
1/4" End Mill	Rough	1.0000																	
<div>3. Select the External parameter.</div>	<div>Router Offset Parameters:</div> <table><tr><td>External (male)</td><td><input checked="" type="checkbox"/></td></tr><tr><td>Internal (female)</td><td><input type="checkbox"/></td></tr><tr><td>Weld offsets</td><td><input type="checkbox"/></td></tr><tr><td>Sharp corners</td><td><input type="checkbox"/></td></tr><tr><td>Inlay</td><td><input type="checkbox"/></td></tr><tr><td>With bridges</td><td><input type="checkbox"/></td></tr></table>	External (male)	<input checked="" type="checkbox"/>	Internal (female)	<input type="checkbox"/>	Weld offsets	<input type="checkbox"/>	Sharp corners	<input type="checkbox"/>	Inlay	<input type="checkbox"/>	With bridges	<input type="checkbox"/>						
External (male)	<input checked="" type="checkbox"/>																		
Internal (female)	<input type="checkbox"/>																		
Weld offsets	<input type="checkbox"/>																		
Sharp corners	<input type="checkbox"/>																		
Inlay	<input type="checkbox"/>																		
With bridges	<input type="checkbox"/>																		
<div>4. In the cut parameters dialog for the 1/4" End Mill tool, change the number of passes to 3.</div> <div>Click OK to get back to the strategy dialog.</div> <div>In the strategy dialog, click OK to create the routing offset toolpaths.</div>	<table><tr><th colspan="3">Passes</th></tr><tr><td>Number</td><td>3</td><td></td></tr><tr><td>Maximum per Pass</td><td>0.7500</td><td>in</td></tr><tr><td>Actual per Pass</td><td>0.3333</td><td>in</td></tr><tr><td>Final Pass</td><td><input type="checkbox"/></td><td></td></tr><tr><td>Final Pass Depth</td><td>0.0000</td><td>in</td></tr></table>	Passes			Number	3		Maximum per Pass	0.7500	in	Actual per Pass	0.3333	in	Final Pass	<input type="checkbox"/>		Final Pass Depth	0.0000	in
Passes																			
Number	3																		
Maximum per Pass	0.7500	in																	
Actual per Pass	0.3333	in																	
Final Pass	<input type="checkbox"/>																		
Final Pass Depth	0.0000	in																	

5. The perspective view is a good way to view the toolpaths.

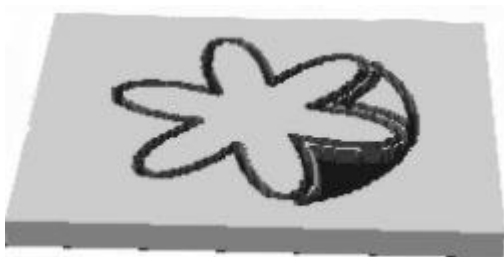


6. This is a rendered view of the final object. It includes the fill and engrave toolpaths from the previous example.



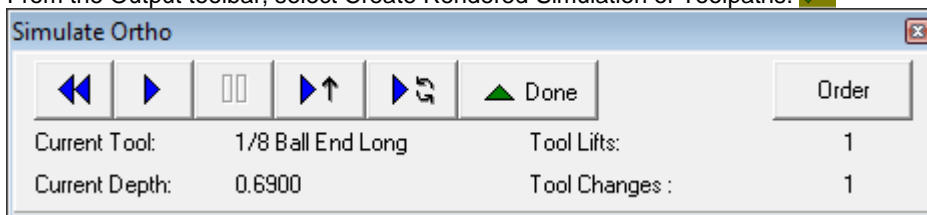
Create Rendered View of Toolpaths

This tool displays a rendered solid model view of the results of the toolpaths cutting the material. It accurately renders all of the different tool shapes, so you can see the results of both 2D and 3D toolpaths.




To simulate output of your design using a rendered view:

1. From the Output toolbar, select Create Rendered Simulation of Toolpaths. 



2. In the **Create Rendered Simulation** dialog, use the controls provided to play the simulation.

 Hit the up and down arrow keys to speed up and slow down the simulation.



Rewind

This button will rewind the simulation to the beginning.



Draw next segment

This will advance the simulation until the next segment has been output.



Pause

This will start the simulation and allow it to play until the end.



Next Lift

This will advance the simulation to the next tool lift.



Next Tool

This will advance the simulation to the next tool change.



End


This will end the simulation and close the Simulate 2D dialog.



Order

This button displays a Toolpath Ordering dialog that is identical to the one in the Output dialog.

Changes in Priority Ordering, Tool, Strategy and Sort Method may all be accomplished directly in the 2D Simulation tool so that the effect of any change may be viewed.

 After any ordering changes are made, be sure to click on the 'Update Order' button so that the changes are reflected in the output and simulation.

3. Click **Done** to end the simulation.

 Be sure to activate the Rendered View icon .

Image with rendered view activated.

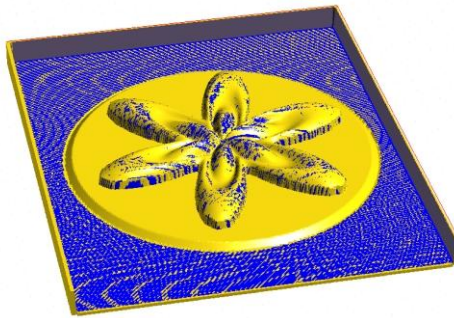
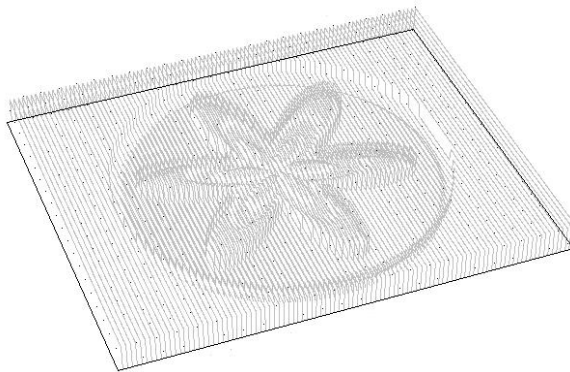


Image without rendered view activated.



22. Automatic Toolpaths

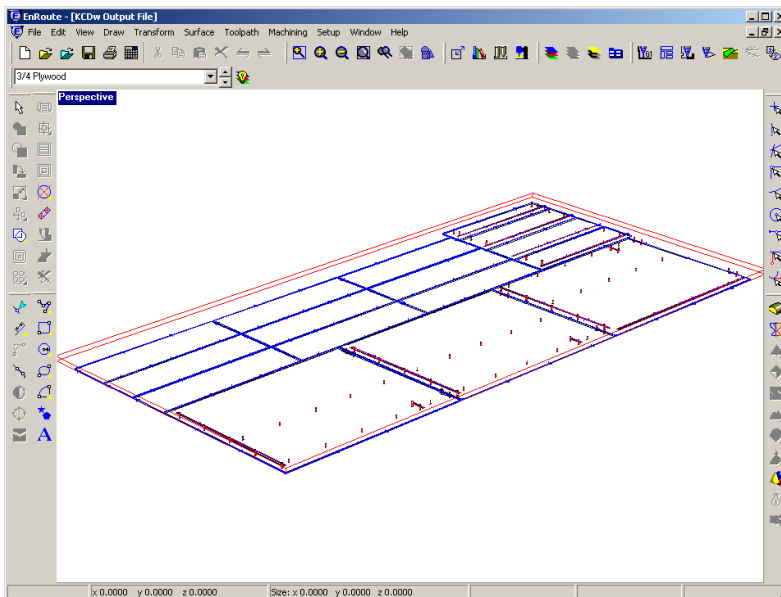
Introduction

The EnRoute Wood version of EnRoute provides Automatic Toolpath (ATP) capabilities that allow you to process parts contained in external DXF geometry files. This type of geometry is commonly generated by design applications that serve industries such as the cabinetmaking and furniture making industries.

EnRoute's Automatic Toolpath capabilities provide an efficient way to process large numbers of parts as part of a Nested-Based Manufacturing (NBM) process.

This chapter provides a description of EnRoute's Automatic Toolpath capabilities, including a description of how to set up the required configuration settings, how to set ordering and output preferences, and then how to process files.

The following graphic shows a sheet of nested parts that have been processed using EnRoute Wood.



Design Application Support

Several design applications are supported by EnRoute Wood, and new application support is added based on requests from users, manufacturers and customers. The following applications are currently supported by EnRoute.

- Cabinet Vision
- Click n' Cut
- Extreme
- FlexiCABS
- KCDw
- Pattern Systems
- Quisine

Editor's Note: The graphics in this Automatic Toolpath discussion must often show a particular design application as the active application. In the interest of simplicity, KCDw was selected as the active application to show in most of these graphics. This is not intended to offer a specific recommendation or to exclude any of the supported applications. References to the active application in the text are either represented as **<active application>** or **<KCDw>** and should be interpreted to mean that any of the design applications can be inserted into these references.

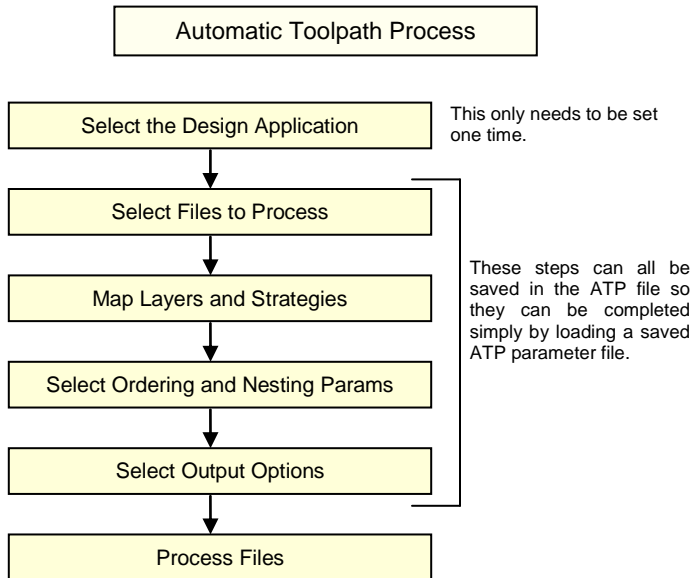
Overview of the ATP Process

When DXF files are created in the design applications supported by EnRoute, different types of machining operations are separated onto different layers in the DXF files. The geometry on all the layers in a file represents a single part, and all of the machining such as drilling, milling, and cutting, that is to be performed on that part. In order to process the part, the ATP needs to be instructed about what operations (Strategies) should be applied to each layer of geometry in each DXF file.

When EnRoute processes a part, it uses this mapping of toolpath Strategies to apply toolpaths to each piece of geometry in the part. After all the parts have been imported and had toolpaths applied, they are nested together onto sheets and then machining output files are created based on the ordering and output preferences defined.

At first, this process may appear to be rather complicated, but the nice part is that after you have defined the desired parameters for Layer/Strategy mapping, ordering and output they can be saved in an ATP configuration file. Then when it is time to process another job the ATP file can be opened to recall all of the desired settings. As few as three mouse clicks can be required to process a job that may contain dozens of parts.

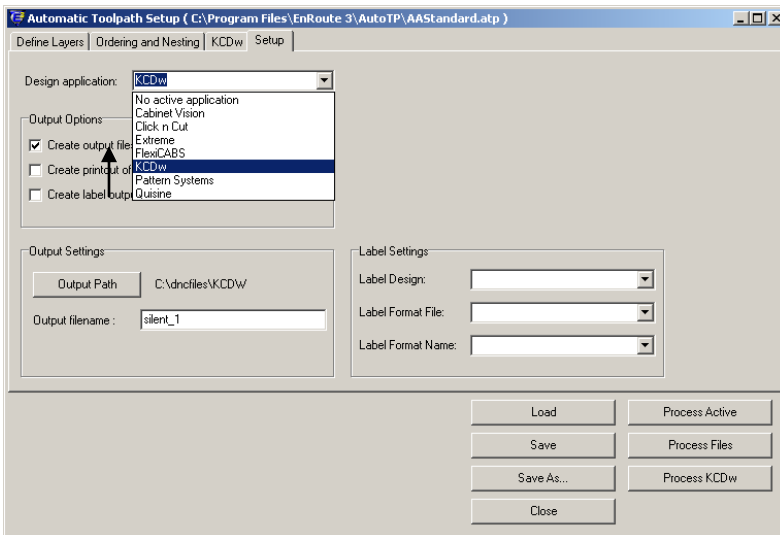
The following diagram shows the general flow of tasks utilized to process an Automatic Toolpath job.



Activating the ATP Dialog

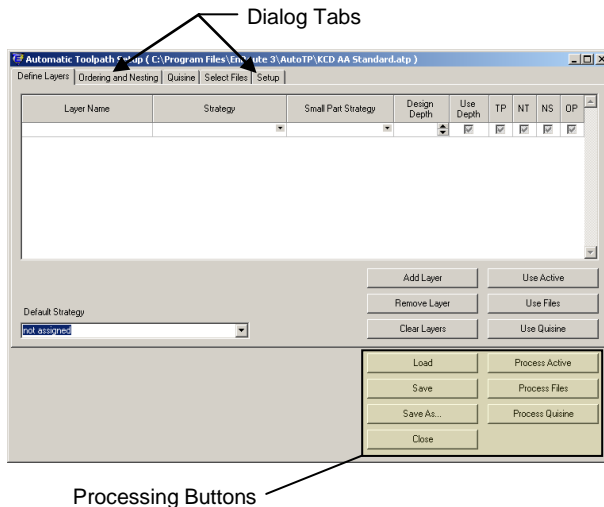


The ATP dialog is activated by selecting the **File** menu and clicking on **Automatic Toolpaths**. It can also be activated by clicking on the Automatic Toolpath button located in the Output toolbar.



The ATP Dialog

All of the ATP functions are contained in one dialog. The tabs in the dialog represent the major steps that were introduced in the ATP overview previously. Initially many of the parameters in the ATP dialog are blank. When a saved ATP parameter file is loaded, this dialog is filled out based on the parameters in the ATP file. The next several sections provide a step-by-step discussion of the process of defining the required layer mapping and other parameters required to successfully process parts using the ATP.



Processing Buttons

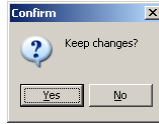
There are seven buttons that are present all the time that the ATP dialog is active. This is because these functions can be activated at any time. These buttons are highlighted above.

Load – This activates a load dialog for loading a saved ATP file. ATP files may be located anywhere that is most convenient the same as standard EnRoute drawing files.

Save – This allows you to save the current parameters into an ATP file. If an ATP file has been previously saved or loaded during this ATP session, the file will be saved without prompting you to identify where, and by what name, to save the file. If the file has not been previously saved, you will be prompted to name the file to save.

Save As – This is the same as the Save button, except that it always activates the save dialog to prompt you for a name under which to save the file.

Close – This closes the current ATP session, and returns you to the EnRoute environment. If changes have been made to current ATP parameters, you will be asked whether to save these changes as the active ATP parameters. If you click on Yes, the current parameters will be saved in EnRoute's memory so that if you get back into the ATP these parameters will be restored.



There are three Process Buttons that tell EnRoute to use the current ATP parameters to process part information. The difference between these buttons is that they each tell EnRoute to process different part information. Very likely you will utilize just one of these buttons for a majority of your work, depending on how your part information is processed.

Process Active – This processes the active EnRoute drawing. This is the drawing that was active when you started the current ATP session. For most users, this option will not be utilized very often, but it can be convenient if you would like to quickly process some parts that you have created in EnRoute, or imported from another application.

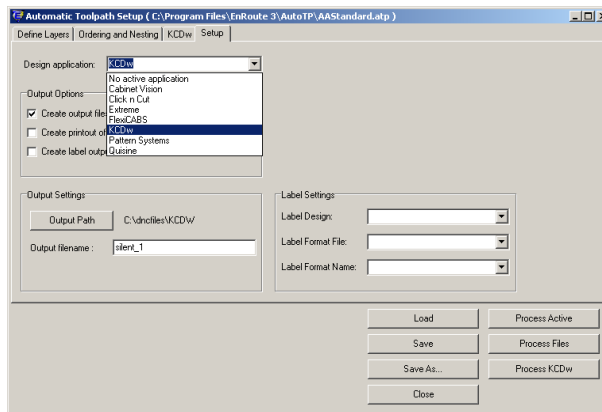
Process Files – This button processes a list of DXF files that you have identified in the list shown in the 'Select Files' tab. This option allows you to process geometry files that have been created by a separate design application that may not be directly supported by the ATP. For example, many users utilize AutoCAD to generate their designs, and then would like to process these designs in EnRoute. This option makes it easy to accomplish this task.

Process <Active Design Application> - This button processes the files that have been specified for the active design application. The button text changes depending on which design application is active. For example, if KCDw is the active design application, the button will read 'Process KCDw'.

Selecting the Active Design Application

If you are using one of the supported design applications, this should be set first before starting the process of defining the layer mapping and processing files. After this has been set, EnRoute will remember this setting so it won't be necessary to define it again unless you change design applications.

To set this parameter, click on the **Setup** tab and select the desired application from the list in the dropdown as shown below.



Note - Notice that it is possible to select **No active application** as an option too. If this option is selected, then individual DXF files may be processed in pretty much the same way as if they came from a supported design application. One key difference is that information about the size, material and orientation of the parts won't be available to interpret as part of the processing.

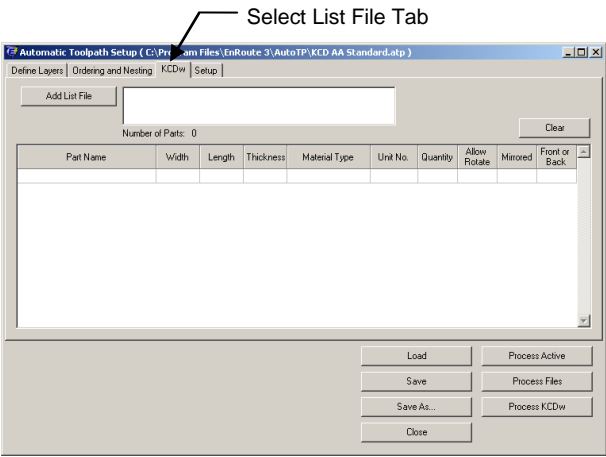
Selecting Files to Process

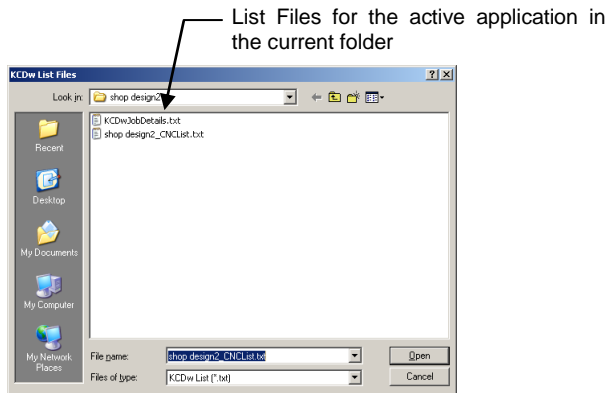
After the design application is specified, the next step is to select which parts are to be processed. The output created by each of the supported design applications includes a *List File* that is a list of all of the separate DXF files in a job.

For example, if you have used KCDw to design a set of kitchen cabinets and then output this geometry to be processed in the ATP, each part is represented by a separate DXF file. The List File for that job is a table that lists all of the DXF files in that job along with additional information such as the size, material, quantity and rotation for each part.

EnRoute allows you to import List Files which load part information for each of the parts in the job. The presence of each of the DXF files is automatically verified, and the information about the parts is shown in the table.

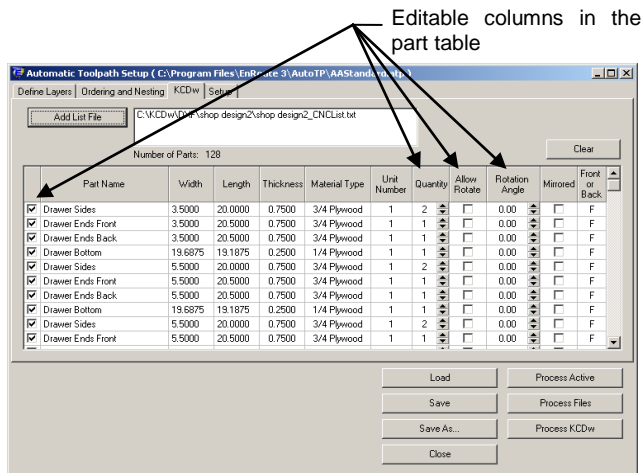
To select a list file, first click on the Design Application tab in the ATP dialog. This tab will have the name of the active design application (Click n' Cut, Quisine, etc...). Then click on the **Add List File** button, which will activate an Open File dialog. This dialog will look specifically for files that match the extension of the list file for the active application.





After the List File loads, the parts are shown in the table. The dialog shows the total number of parts to be processed. If you process a job after a List File has been imported, the parts will be created just as they were specified. As an option, some of the parameters about the parts may be modified if desired. It is not possible to change such things as the size of the part, but you can change the quantity of each part, turn off the processing of selected parts, or custom rotate individual parts.

The following screen capture shows the table after it has been populated with the parts from a selected List File. One thing to note is that the parameters included in this table will change depending on the active design application. This is controlled by the information that is provided by each application.



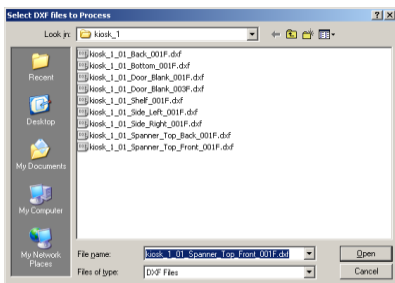
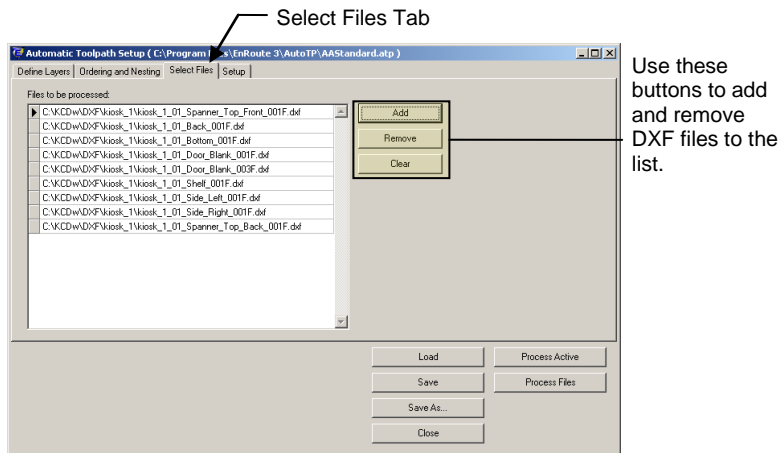
It is possible to process multiple jobs simply by adding more than one list file to the list of parts. If a list file is included in the list that shouldn't be, click on the **Clear** button to clear the parts list and then add the appropriate parts to the list.

When ATP parameter files are saved, the part information in this tab is not saved as part of that file. This is because this information represents project-specific information that will change with each job.

Note again that the Process buttons remain visible when this tab is active. If all of the other parameters have been specified correctly, all that is necessary now is to click on the **Process <KCDw>** button to complete the job. If the ATP parameters have not been defined, the next step is to move on to the Layer Mapping step which is described in the next section.

Selecting Files with No Active Application

In the case where there is not an active design application, part files are selected individually rather than by using a List File. Click on the **Select Files** tab, and then click on **Add** to activate an **Open Files** dialog. Find the DXF files to be processed and select them and click on the **Open** button and they will be added to the parts list in the ATP.



Layer Mapping

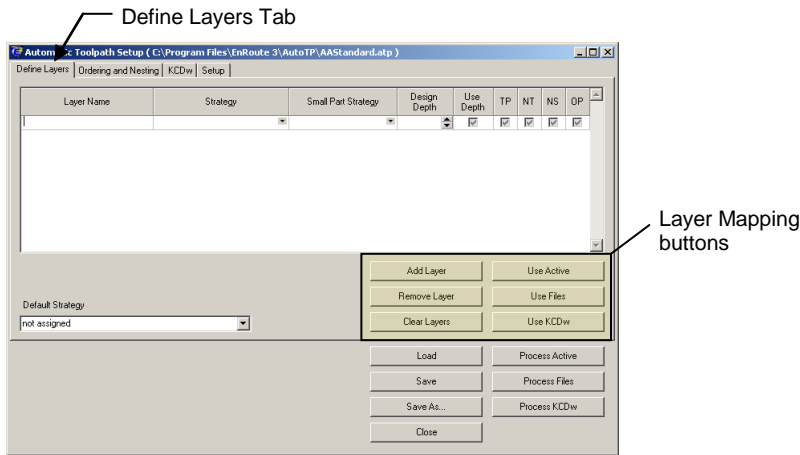
This is a very important step in the process, and essentially represents the core concept in Automatic Toolpath processing. When design application jobs are created, DXF files are utilized to pass on geometry and other design information. Required machining operations will likely include routing offsets, drills and fills, using a number of different tools and depths.

The geometry in the DXF files is separated onto different layers based on what type of machining will be required. Shapes to be cut out will go on a layer, geometry for dados will go on different layers, and drill points on different layers. This arrangement of geometry on specific layers can be customized to match very specific manufacturing methods, but after it has been specified for a given

method it likely won't change.

The Layer Mapping in EnRoute's ATP provides a method to synchronize the toolpath creation to match the geometry in the DXF files. As an example, if the rectangle that represents the perimeter of a part is placed on a layer called *Cutout*, the ATP should be configured to look for the layer *Cutout* and then to create a routing offset toolpath using the desired EnRoute Strategy. This is where that happens.

First activate the **Define Layers** tab by clicking on it. The layers table will initially be blank. What is needed is to create a list of layers that corresponds to the layers included in the DXF files of the jobs to be processed. Then Strategies will be assigned to these layers. When the job is processed, these Strategies will be applied to all of the geometry that resides in that layer in each of the DXF files.



Define the Layers

The Layer Mapping buttons provide the means to create and edit the layer definitions.

Add Layer – Click this button to add a new blank layer to the list.

Remove Layer – Click this button to remove the selected layer.

Clear Layers – Click this button to remove all the layers in the list

One method of defining the layers is to add layers one at a time and then edit the Layer Name to match a layer in the DXF files. In order to edit the layer name, just click in the row of the Layer Name column to be edited and type. Continue to add layers and edit them until all of the desired layers have been added.

This can be a complicated process because you may not know all the layer names that are included in the job to be processed. One solution is to import the DXF files into EnRoute in order to find all the layers. A better alternative is to use the 'Use' buttons. If a List File has been specified so that a list of parts is active, it is only necessary to click on the **Use <KCDw>** button to find the layers to map. EnRoute will automatically open each of the part files and extract the layers included and then add these layers to the layer-mapping table.

Use Active – Click this button to extract all the layers in the active EnRoute drawing.

Use Files – Click this button to extract the layers in the list of individual files that have been selected in the case where no design application is active.

Use <KCDw> - Click this button to extract the layers from the DXF files that represent the parts that have been selected in the parts list. If no parts have been selected, then this button won't add any new layers.

Click the *Use KCDw* button to bring in the layers from the active parts list

At this stage, the Strategies have not been mapped to the layers.

Layer Name	Strategy	Small Part Strategy	Design Depth	Use Depth	TP	NT	NS	OP
PANEL	not assigned	not assigned	0.0000					
ROUTDRAWER	not assigned	not assigned	0.0000					
DRILLPILOT	not assigned	not assigned	0.0000					
DRILLDRAWER	not assigned	not assigned	0.0000					
ROUTBACK	not assigned	not assigned	0.0000					
ROUT	not assigned	not assigned	0.0000					
ROUTTAB	not assigned	not assigned	0.0000					
ROUTBACKTAB	not assigned	not assigned	0.0000					

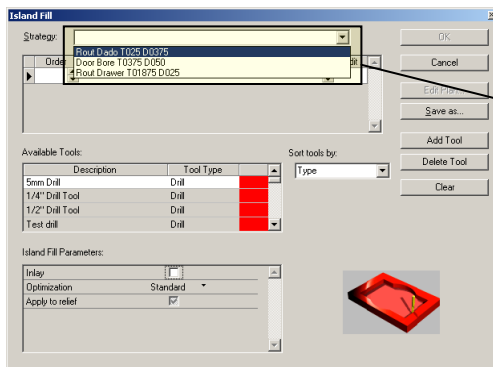
Map Strategies

The next step is to select the Strategies to assign to each layer. Remember that strategy templates are saved using the Strategy dialog in EnRoute. Strategy templates can be saved for every type of toolpath that can be created by EnRoute. After a Strategy Template has been saved, it is then available to the ATP to be specified in the ATP Strategy Mapping table.

Click the **Save as...** button in the Strategy dialog to save the active Strategy as a template. This saved template can then be used by the ATP

Router Offset Parameters:

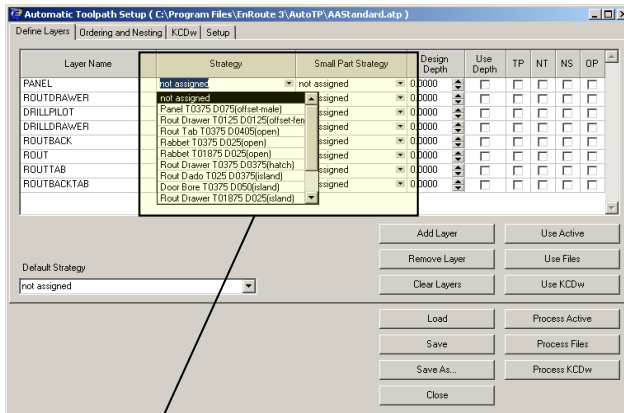
- External (male) ☒
- Internal (female) ☐
- Weld offsets ☐
- Sharp corners ☐
- Inlay ☐
- With bridges ☐



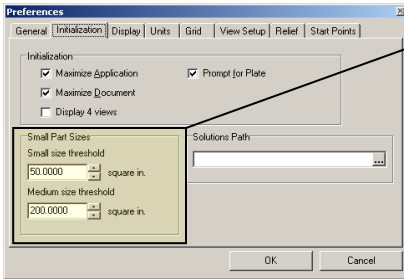
Use the Strategy template dropdown to review the names of the saved strategies, and to select a template to activate.

In the ATP dialog, the saved Strategy templates are selected for mapping to each layer in the list. Both a normal **Strategy** and a **Small Part Strategy** may be selected for each mapped layer. The Small Part Strategy will be used for any parts that have a surface area that is less than a threshold defined by the user in the EnRoute **Preferences** dialog.

In order to select a strategy in a layer, just click on the dropdown arrow on the right side of each strategy cell. This will display a list of all eligible saved strategies. The type of strategy is shown in parentheses next to each strategy name. For example, in the following screen capture, the first strategy in the list is '*Panel T0375 D075(offset-male)*' which tells us that this is a male routing offset strategy. In this case, the name of the strategy was used to provide some information about the tool and depth, and its name was created to match the name of the layer it is to be assigned to. There are many different strategy naming conventions that can be used, and each user should arrive at a system that works best for their situation.

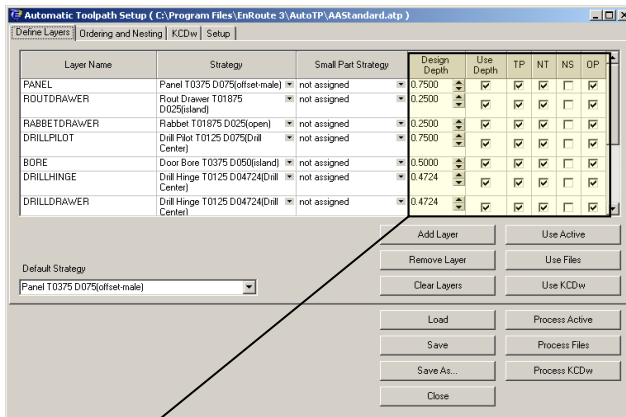


Both a normal **Strategy** and a **Small Part Strategy** may be defined. The Small Part Strategy is utilized for parts that have a surface area below a threshold that is defined by the user. If a Small Part strategy is not defined, then the Strategy is used for all the parts.



Activate the Preferences dialog by clicking **Setup** and **Preferences...** Then click on the **Initialization** tab to show the area threshold for small parts.

After strategies have been mapped to each of the layers, it is necessary to define several additional parameters in order to complete the Layer Mapping task. These include Design Depth/Use Depth, TP, NT, NS and OP. Although the representation of these parameters is a little cryptic, the settings will likely remain the same for most of your jobs.



Additional parameters in addition to the Strategies that must be defined for each mapped layer.

Design Depth/Use Depth – This parameter provides a way to utilize any given Strategy template for more than one cut depth or thickness of material. 'Design Depth' refers to the depth that the selected Strategy was configured for.

When Strategy Templates are created, the strategy is specified with a specific depth. If a routing offset depth is set to a depth of 0.50 inches, then this is the depth at which toolpaths will be created when this strategy is applied. If this strategy is used to cut out material that is 0.50 inches thick, then this works great. However, if I also want to use this same strategy to cut out material that is 0.75 inches thick, there could be a problem. The **Design Depth** parameter helps solve the problem. In this case, I would set Design Depth to 0.50 inches, and click on **Use Depth** to put a check in the checkbox.

When the DXF geometry is created by the design application, it is typically placed at the depth at which it is intended to cut. Contours that are to be cut at 0.50 inches are placed 0.50 inches below zero in the vertical (Z) axis. If **Design Depth** is specified, and **Use Depth** is checked, EnRoute uses the vertical position of the contour to determine whether to automatically adjust the Strategy depth.

In our example, if the contour is at -0.50, then no changes are made, but if the contour is at say -0.75, then the strategy would be modified to have a depth of 0.75. This means that the same strategy could be assigned to layers that will import geometry at different depths.

Some good examples of this include dado layers that are intended to cut at different depths, and cutout layers that may be used for cutting 0.25, 0.50 and 0.75 inch thick material. The advantage to this method is that it ultimately requires fewer strategy definitions to accommodate the range of toolpaths that may need to be created on a job.

When Strategy templates are saved, it may be a good practice to include the design depth in the name of the strategy so that it is easy to remember this number as it is utilized in the ATP.

TP – Toolpath. This provides an option for creating toolpaths for the geometry on each layer. Most of the time this parameter should be checked. There may be cases, however, where information on a particular layer should not get toolpaths, but it should be included in the processed files.

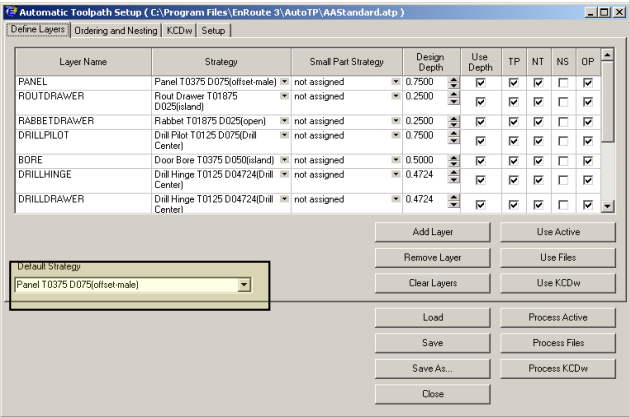
NT – Nest Together. This tells the ATP that this layer should be included with the 'part' that is defined in the rest of the DXF file. When output from a supported design application is being processed, this parameter should always be selected. There may be certain situations when DXF files that were created some other way are being processed. In these cases it is possible that several parts may be included in one file, and this is the situation that may work better using the NS (Nest Separate) option.

NS – Nest Separate. This option tells the ATP to treat individual contours as separate parts, and to not treat each DXF file as a complete part. See the discussion for the NT option for the situations when this may be appropriate.

OP – Output. The layer should be included with the part in the toolpath output. This should be checked in virtually all cases. There may be unique situations in which geometry information is included in the part files that should not be included in the output.

Default Strategy

The **Default Strategy** provides an option for specifying a Strategy to be utilized in case a layer is included in the processed parts that is not included in the layer mapping. It is not necessary to specify the Default Strategy.



Ordering and Nesting

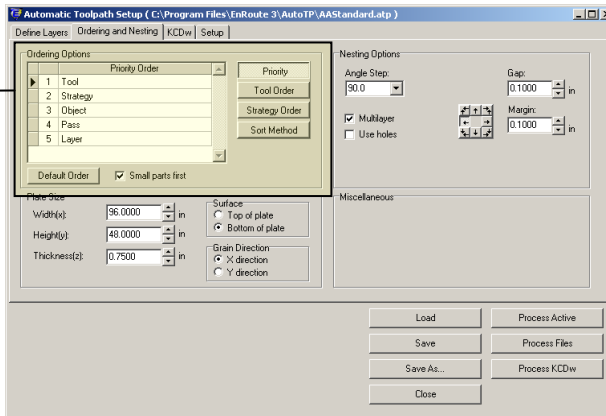
After the layer mapping is complete, then it is necessary to specify how the toolpaths should be ordered, and then how the parts should be nested. This part of the process is virtually the same as the ordering and nesting that is performed within the normal EnRoute output process.

To activate this tab, click on the **Ordering and Nesting** tab in the ATP dialog.

Ordering Options

The ordering options are the same as within EnRoute. A detailed description of this is included in Chapter 10. The following provides a description in the context of the ATP.

The **Ordering Options** provide the means of specifying the order of the toolpaths when the output files are created.



Priority

The Priority determines the order in which the toolpaths are ordered, using each of the five Priority options. These items may be placed in the desired order by clicking on the item to be moved, and then clicking and dragging on the item to move it.

When the output is created, the toolpaths are ordered working from the bottom item in the list up to the top Priority item. For a given output file, the toolpaths are ordered five times; so, for the list in the dialog above they would be ordered first based on the order of the Layers, and then based on the Pass depths, then based on the sorted Objects, then based on the Strategy Order, and finally based on the Tool Order.

The result of this process is that the toolpaths will always be ordered to match the order of the top Priority item. They will be ordered to match the order of the second Priority item to the extent that it is consistent with the Tool Order, and so on down the Priority list. This means that the top items have the most impact on the order of the toolpaths.

The following simple example illustrates the concept of the **Priority Ordering** of toolpaths using numbers to represent toolpaths.

The following numbers are unordered. The numbers aren't repeated, and they are either big numbers or small.

5, **8**, 3, 9, **2**, 7, **1**, **4**, 6

The numbers are first ordered based on their numeric sequence.

1, **2**, 3, **4**, 5, 6, 7, **8**, 9

Now, the numbers are ordered based on size, with the small numbers first, providing the following final order.

3, 5, 6, 7, 9, **1**, **2**, **4**, **8**

You can see that by ordering by numeric sequence first and then size we get a specific order to the numbers. Within each of the two size groups, the numbers are in numeric sequence even though they are not in numeric sequence overall.

In the second option the numbers are first ordered based on their size.

8, **2**, **1**, **4**, 5, 3, 9, 7, 6

Now, the numbers are ordered based on numeric sequence, providing the following final order.

1, **2**, 3, **4**, 5, 6, 7, **8**, 9

You can see that by ordering by size first and then numeric sequence we get an order that is quite different than in the first option even though all we did was switch our ordering priorities.

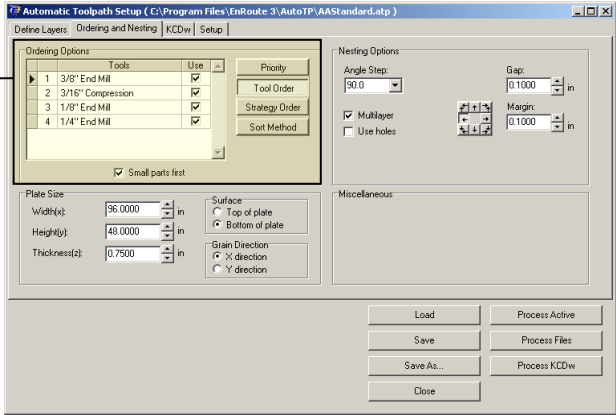
Tool Order

The Tool Order gives the opportunity to define the order of the tools that are used in the current job. This order is used when the toolpaths are ordered using the Tool priority. EnRoute automatically lists the tools that are utilized in the Strategies included in both the Layer Mapping and the current parts to be processed.

Note that EnRoute checks both the parts in the parts list as well as the Strategies in the layer mapping for the tools. This means that if you have not yet specified any parts to process, there won't be any tools in this list. As soon as parts are selected to process, the required tools will show up in the tool list.

Tools can be included or excluded from the output by checking or un-checking the **Use** checkbox for each tool.

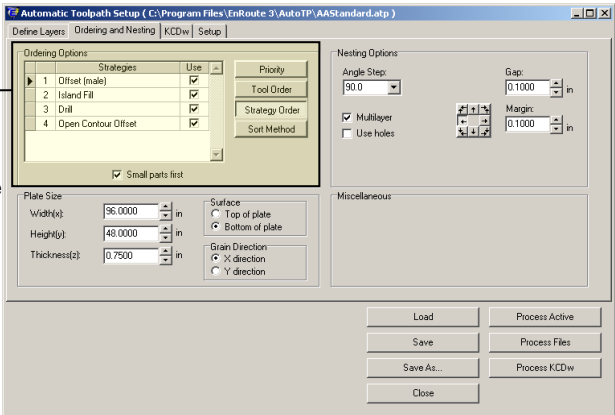
Tool Order shows the order of the tools required by the current Layer Mapping and parts list. The tools can be moved in the list to the desired tool output order.



Strategy Order

The Strategy Order is similar to the Tool Order. The Strategy list is populated using the strategies from the layer mapping, for those parts that are included in the parts list. This order is used when the toolpaths are ordered using the Strategy priority. A strategy can be excluded from the output by unchecking the **Use** checkbox for that strategy.

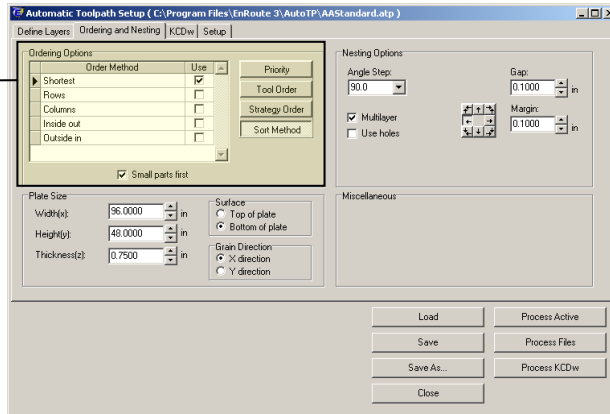
Strategy Order shows the order of the strategies required by the current Layer Mapping and Parts list. The strategies can be moved in the list to the desired output order.



Sort Method

The Sort Method defines how *parts* on a sheet will be sorted. This part order is used when the *toolpaths* are ordered using the Object priority.

Sort Method lets you select how the parts on each sheet will be sorted. Typically you will use the Shortest method, but you may occasionally prefer to sort the parts into Rows or Columns.



Small Parts First – This checkbox lets you ensure that the parts will be sorted so that any parts that are smaller in area than the Small Part threshold will be ordered before the larger parts. This option can be particularly useful if you are using a vacuum system to hold parts down. It allows the small parts to be cut out first, when the vacuum is most capable of holding them down.

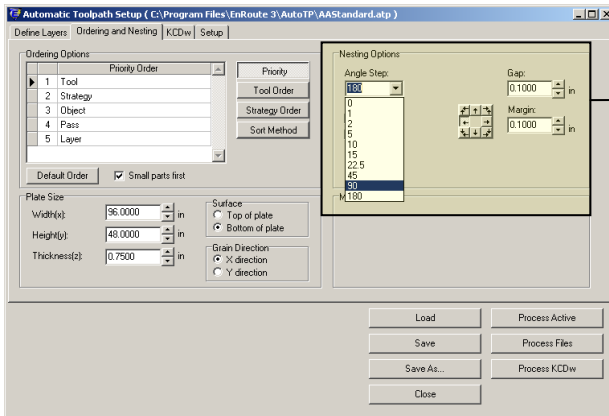
Nesting Options

The Nesting Options provide the parameters to specify how the parts should automatically be arranged on the sheets as they are prepared for output. The nesting process attempts to make the most efficient use of the material, minimizing waste.

EnRoute and EnRoute Wood use a true-shape nester, which means that it considers the precise shape of the part instead of treating the part as a rectangle in all cases. The advantage of true-shape nesting is that it can make the best use of the material by arranging parts so that waste is minimized.

Nesting of the parts is a key step, because the nested sheets are the basis for the output files that are created. When the parts are processed, an EnRoute drawing is created that contains layers that represent each of the nested sheets. Each drawing layer corresponds to an output file. In the nested sheet drawing, the layer names correspond to the material that sheet represents.

During the nesting process, the ATP keeps parts that are to be cut from the same material together. This means that if you have a job that includes $\frac{3}{4}$ " melamine, $\frac{1}{4}$ " plywood, and $\frac{1}{2}$ " plywood, then all of the $\frac{3}{4}$ " melamine parts will be nested together, all of the $\frac{1}{4}$ " plywood parts will be nested together, and all of the $\frac{1}{2}$ " plywood parts will stay together. EnRoute uses the material names included in the List Files to determine what material each part will be cut from.



Nesting Options provides the parameters for EnRoute's true-shape nester to arrange parts on the sheets.

Angle Step – These are the angle increments that the nester uses for each part as it works to find the best arrangement of the parts on the sheet. Smaller angles increase the number of options that the nester must consider, so it increases the amount of time to nest each sheet.

When processing files in the ATP, much of the material to be cut has a wood or some other directional surface so often times it is best to set the step angle to 180 degrees. This minimizes the options for the nester to consider, and will help reduce processing time. Even when the material is not grained, experience shows that setting the step angle to some value below 90 degrees most often is not necessary in order to make the best use of the material.

Gap – This is the spacing between the individual parts. The overall size of each part includes the size required by any external toolpaths, so it is often acceptable to set the Gap to a very small value, or even 0.0, in order to maximize material usage.

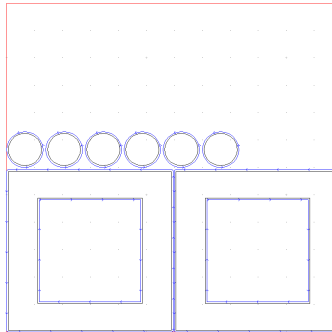
Margin – This is the spacing from the edge of the sheet of material to the parts nearest the edge. Again, this value may be set to a small value since the boundary of each part includes the toolpath dimension.

Multilayer – This tells EnRoute that it can create new sheet layers in the output drawing as necessary in order to nest all the parts. For example if there are 50 parts to be cut from $\frac{3}{4}$ " melamine and only 10 parts will fit on each sheet, EnRoute will automatically create the 5 sheet layers in the output drawing required to nest all the parts. Consecutive sheet layers are named to show how many sheets are required of each material.

Use Holes – This allows the nester to utilize the holes in which to nest parts on a sheet. This option does not typically get used with the files that are most commonly processed in the ATP. The following graphic shows the difference between nesting with and without the Use Holes option checked.

Use Hole is Unchecked -

The rectangles have an inner 'hole' that could potentially be used for cutting parts. If Use Holes is not checked, then objects are not placed in these holes.



Use Hole is Checked - In

this case you can see that the holes are utilized for placing objects. The result is that it can be possible to cut the same number of parts by using less material.

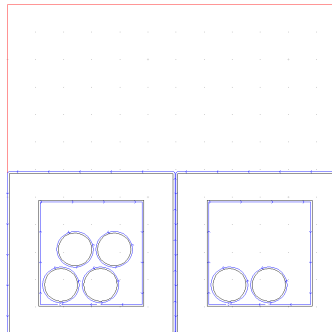


Plate Size

This section tells the ATP what size material is going to be used for cutting the job, and where surface is to be defined.

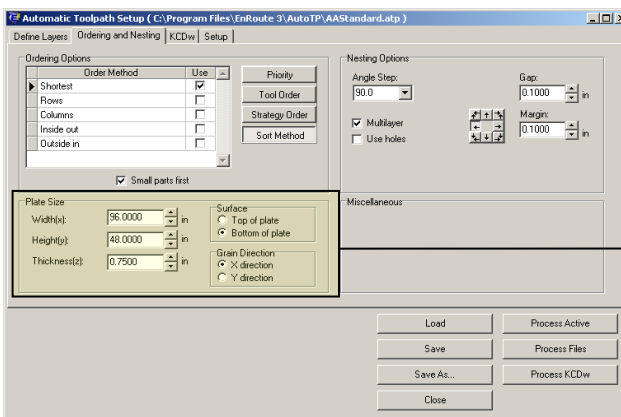


Plate Size tells the ATP what size material will be used for cutting the parts, and where the surface will be defined on the machine.

Width – This is the X dimension of the material. The units of this dimension match the active units in EnRoute.

Height – This is the Y dimension of the material.

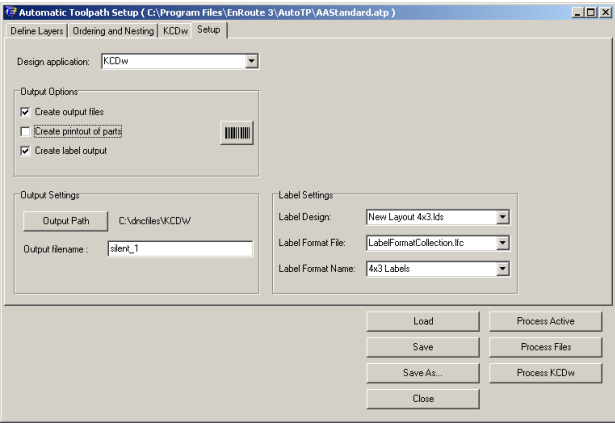
Thickness – This is the default Z dimension of the material. When output is created for the active design application, the plate thickness is automatically adjusted as necessary to match the thickness of the parts on each sheet. If files are being processed with no active design application, or if the active EnRoute drawing is being processed then this thickness is used.

Surface – This is a user preference that may change depending on the machine being used, or the type of material being cut. Typically the surface is set to the bottom of the plate because then small variances in material thickness do not cause problems when cutting parts out.

Grain Direction – The assumption is that parts coming into the ATP are oriented based on the grain direction in the X direction. If this parameter is set to the Y direction, then parts are automatically rotated by 90 degrees as they are imported from the DXF files.

ATP Setup

The Setup tab contains parameters that determine what type of output files are created when a job is processed. There are three types of output that can be generated, including (1) the machining output files (the g-code files), (2) a printout of each nested sheet that shows what parts are included on each sheet, and (3) label output files that allow labels to be printed for each part.

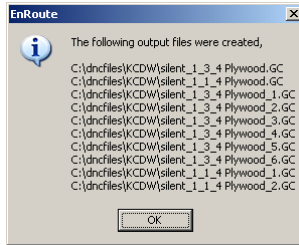


Output Options

These checkboxes allow you to choose what output you would like to generate.

Create output files – This option creates the machine files that will be sent to the router. The files will be located in the location that is specified in the Output Settings section described below.

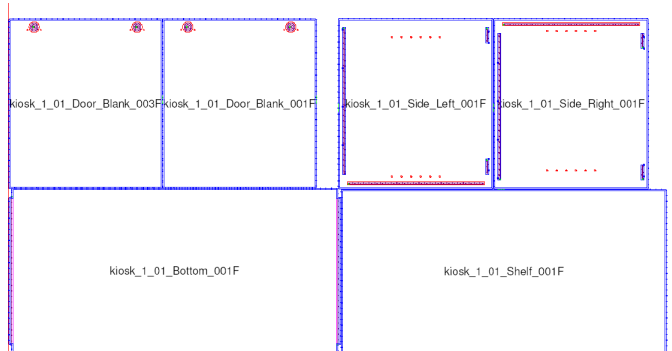
When a job is finished processing the ATP provides a message box that lists the output files created.



Create printout of parts – This option prints an image of each nested sheet, including a text label that identifies each part on the sheet, as well as a title that is the name of the output file that was created. This type of printout makes it much easier to identify individual parts so that they can be organized as they come off the machine after they are cut.

The barcode button to the right of this checkbox allows you to specify a barcode font that will be used to generate a barcode version of the filename of the output file. Some machine manufacturers allow their users to locate and track output files using barcodes and this helps facilitate that option.

The following image shows an example of a printout of a job processed in the ATP.



Create label output – This tells EnRoute to generate label output using the parameters defined in the Label Settings on this dialog. Labels provide a higher level of tracking ability for the parts. This is an option that can be added to the standard EnRoute Wood package that allows you to design and generate individual labels for each part that you process in the ATP. Labels can form a key part of a manufacturing process. EnRoute uses an XML format that makes it easy for design applications to provide a wide range of information to EnRoute that can be passed along to each label. Information about edge banding, customer, finishes, or virtually any other information can be included in labels.

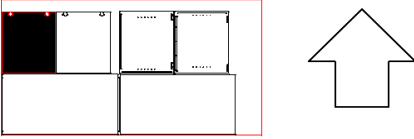
Following is an example label. It was designed to include standard information about the filename, dimensions and material. It also includes a thumbnail view of the nested sheet that shows which part this label is for, along with an arrow that indicates the orientation of the part on the sheet.

Filename : kiosk_1_01_Door_Blank_003F.dxf

Dimensions: 21.825 X 19.825 X .75

Name: Door Blank

Material: 3/4 Plywood

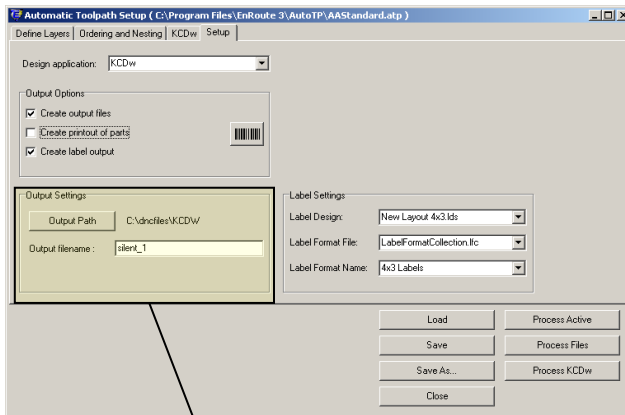


Output Settings

These two parameters tell EnRoute where the output files should be placed, and how they should be named. The **Output Path** button activates a dialog for specifying the path to the output files. This path is displayed to the right of the button.

The **Output filename** parameter serves as a prefix to the names of the output files. It is a way to define filenames that are easily identifiable. Output files are automatically named using the material name for the parts. If the Output filename parameter is set, then the material name portion of the filename is appended after this parameter.

As an example, if the Output filename parameter was set to 'Smith01' and the material it was to be cut using is '3/4 Plywood' then the output filename would be 'Smith01_3_4 Plywood.cnc'. Notice that the '/' is replaced with '_' since this is not a legal character in a filename. Also, the extension on the filename would be based on the active machine.



The Output Settings parameters allow you to specify where output files are saved, and a prefix to the file names.

Label Settings

If the labeling option is enabled, these parameters are used to select the size and design of the labels to be printed. Two separate applications are used to define Label Design and Label Format files.

Label Design – This file format uses the XML file format to specify the layout of the labels. The files have an 'lds' extension and are created using EnRoute's **Label Designer™** application.

Label Format File – The label formats specify how labels are arranged on a sheet. Many standard label formats are provided with EnRoute, and new ones are easily created using the **Label Maker™** application. The 'lfc' file format stores the label format, and is selected with this parameter.

Label Format – When the Label Format file is specified, its individual label formats may be selected using this parameter.

Example Steps to Process Your Files

The previous sections describe each of the steps required to use EnRoute's Automatic Toolpath feature to process part files. This section provides a brief step-by-step listing of how you would typically proceed through the process. It follows the flowchart described in the ATP Overview presented at the start of this chapter.

Processing a Job Starting from Scratch

These steps assume that you are setting up the ATP from the start, and have not yet performed any Layer Mapping and have not set any of the other parameters in the ATP.

1. Start EnRoute.	
2. Define strategy templates in EnRoute based on the machining operations and tools you typically use.	If you are a new EnRoute user, this will likely take some trial and error on your part to determine the machining operations you will typically use.
3. Set up the EnRoute Tool Library to include the tools you will use with your ATP machining.	EnRoute is supplied with a default tool library that you will likely modify to meet your needs.
4. Configure the Machine Setup parameters in EnRoute so that they match your actual machine configuration.	This will likely include configuring your tool changer and/or drill bank to match your machine configuration.
5. Start the ATP.	You are now ready to set up the ATP.
6. Go to the Setup tab and set the Active Design Application.	
7. Go to the tab for the active application (KCDw, etc...) and select the File	Prior to this you should have created job output from your design application.

List for the job you would like to process.	
8. Go to the Define Layers tab in order to define the Layer Mapping.	Now we need to set the layers and strategies we will use to process the job.
9. Click on the Use KCDw (or other application) button in order to find what layers are used.	This will extract all of the layer names from the DXF files in the active job so that you don't have to search them out yourself.
10. Select a Strategy for each of the layers.	These will be used to process the geometry on the corresponding layers in the parts.
11. Select a Small Part Strategy for the cutout layer(s) if that is desired.	Remember that you need to define the small part threshold in EnRoute's preferences if you are going to use this.
12. Define the Design Depth/Use Depth parameter for each layer.	If you would like EnRoute to automatically adjust toolpath depths based on the geometry depth in the DXF files, define the depth that the strategy was designed for and then check Use Depth.
13. Select TP, NT and OP for each layer.	These parameters should be checked for virtually all standard applications.
14. Define a Default Strategy if you want one.	This probably isn't necessary for standard jobs, and it could get you in trouble if a surprise layer shows up in a file.
15. Click on the Ordering and Nesting tab	
16. Set the Priority order you want to use.	You likely want to have either Tool or Strategy at the top of the Priority list.
17. Define the Tool Order.	EnRoute extracts the tool list based on your Layer Mapping. Generally, move tools used for drilling and fills ahead of tools used for routing offsets.
18. Define the Strategy Order.	These are the strategies that the Layer Mapping uses. Typically you want to have the male routing offsets at the end so that objects don't cut out before they have finished the other operations.
19. Select the Sort Method.	Typically the Shortest option works fine.
20. Set the Plate Size.	What size material are you going to use? 96x48, 97x 49 etc... If you are processing output from a design application then the thickness parameter will be adjusted automatically.
21. Set the Surface option.	Do you typically set the machine surface at the top or bottom of your material?
22. Set the Grain Direction.	Typically leave it in the X direction.
23. Set the Nesting Options.	Angle Step = 180 or 90

	<p>Multilayer = checked</p> <p>Use Holes = unchecked</p> <p>Gap = 0.0 or some small number</p> <p>Margin = 0.0 or some small number</p> <p>Position Button = Bottom, Left or Lower Left typically.</p>
24. Click on the Setup tab.	
25. Set the Output Options.	<p>Check the appropriate options. If this is the first time through, you may not want to generate any output until you have reviewed a few jobs. Because EnRoute creates an active drawing containing all the nested sheets it is easy to review the results of the processed job without having any other output created. Just scroll through the layers to see the nested sheets.</p>
26. Set the Output Settings.	Tell EnRoute where to send the output.
27. Set the Label Settings.	If you are creating label output, choose a label design and a label format. Remember that labeling is an optional feature.
28. SAVE THE ATP FILE.	Before you do any processing, click on the Save button and save all of the setting you have defined. Then when you need to make changes you will only have to change the parameters that need to be modified.
29. Click on the Process <KCDw> button.	The button will have the name of whichever design application you are using.
30. Click on the Close button and review the results of the job in the active EnRoute Drawing.	You will be prompted to Save Changes. If you click on Yes then all of the parameters and the list of parts will be retained so that if you get back into the ATP things will be as you left them.

Processing a Job Using a Saved ATP File

This example assumes that you have already defined the job parameters and now need to process a new set of part files. It illustrates that after you have been through the process of mapping the layers and setting the other processing parameters it becomes very quick and easy to process jobs that can contain dozens of parts.

Open the ATP.	
Load the List File you want to process.	
Load the saved ATP file.	
Click on Process <KCDw>.	
Review the nested sheets.	Flip through the sheets in the EnRoute drawing to make sure things look correct.
Send the output files to your machine.	

Keyboard Shortcuts

Bump by Bump Increment	Arrow Keys	Preferences	F10
Bump By Display Percentage	Shift + Arrow Key	Redo	Ctrl+ Shift + Z
Copy	Ctrl + C	Redraw	Ctrl + R
Cut	Ctrl + X	Shortcut Menu	Right Click
Define Layers	F7	Rotate	Shift + F7
Delete	Del	Save	Ctrl + S
Exit	Alt +X	Scale	Ctrl + F6
Group	Ctrl + G	Select All	Ctrl +A
Help	F1	Simulate 2D	F8
Machine Setup	F6	Tool Library	F5
Material Library	F4	Undo	Ctrl + Z
Move	Shift + F5	Ungroup	Ctrl + U
New	Ctrl + N	View Toolpath Thickness	F9
Open	Ctrl +O	Zoom In	Ctrl + I
Paste	Ctrl + V	Zoom Out	Ctrl + K
Perspective / Top View Toggle	F12	Zoom Plate	Ctrl + P
Precision Input Center	F2		

Index

2D output simulation	289	Bitmaps	285
3D Meshes	261	Bridges	128
apply meshes to relief	264	deleting	167
create mesh with EnRoute tools	262	displaying	22
faceted	264	editing	165
import 3D mesh objects	262	inserting	166
smooth	264	Build Parameter	299
using a mask	266	Calculator	55
3D Surfaces		Carve into surface	302
Introduction	197	Centerline	250
3D Surfaces Toolbar	34	Chamfer Tool	247
3D Toolpaths	135, 197, 301, 311	baroque carve	258
3D toolpath example	307	centerline dialog	255
3D toolpath strategies	301	chamfer centerline	254
3D toolpath strategy parameters	302	chamfer example	257
engrave example	318	chamfer parameters	250
finish pass example	316	miter and centerline	250
key 3D toolpath concepts	305	standard chamfer	247
overlap	306	standard dialog	249
roughing pass example	314	wizard prompts	252
routing offset example	320	Chamfer Toolbar	38
step rough	304	Changing cut type	131
step rough example	311	Circles	
Aborting a job	195	by center and point	65
Active drivers	178	by center and radius	65
Align Reliefs Toolbar	38	by corners	66
Aligning Contours	114	by three points	66
Alignment Toolbar	36	Clean cut	140
Aluminum	130	Cleaning Pass	140
Angles for Revolution	215	Cleaning Up Contours	95
Apparent intersections	81	Climbing cuts	134
Apply to Surface	302	Communication parameters	177
Arc Copy	90	Configuring Machine Drivers	175
Arcs		Construction Toolbar	33
by Center, Radius and Angle	67	Containers and holes	97
by center, start and finish	67	Contour Editing Toolbar	33
by three points	68	Contour loop indicators	22
converting to arcs	125	Contours	
copying around an arc	90	closed	126
editing	78	displaying	22
Array Copy	89	editing	77
Automatic cleanup	98	extending	79
Bezier curves		filleting	83, 84
converting to arcs	125	merging	98
drawing	62	merging open contours	87
editing	78	selecting	97
extending	79	trimming	81
trimming	81	Convert Curves to Arcs	88

Copy	
array copy tool.....	89
copy around and arc	90
path copy	92
Corner tags.....	145
Corners	
corner tags	145
filleting	83, 84
sharp	127
Create Rendered View	172
Creating new files	51
Cut by Line Tool.....	118
Cut Definitions	
offset from surface	303
Cuts.....	125
changing cut type	131
clean	131, 138
climbing	134
cut parameters	130
bridges	166
engraving	158
entry/exit points	164
hatch fill	142
island fill	145
open contour offset	138
pyramid	161
cut templates.....	126, 135
determining cutting order	189
direction	134
dwell	133
entry/exit parameters	134
fill	138
fine	138
passes	132
plunge rate	133
rough	131
shoulder.....	133
spindle speed	134
width	132
Cutting order.....	189
Deleting	
bridges.....	167
jobs	195
strategies.....	162
Deleting Contours.....	98
Depth of cut	127, 131, 136, 140
Depth of toolpaths.....	23
Designs	
closing	54
creating	51
drawing.....	59
exporting	54
importing.....	54
opening	53
saving	53
Dimension Toolbar.....	39
Dimensions.....	73
angle dimension tool	74
diameter dimension tool.....	75
leader dimension tool	75
radius dimension tool	74
single dimension	74
Direction arrows	22
Direction arrows on toolpaths.....	22
Distort Contours	110
Done queue.....	194
Draft Angle.....	276
Draw Lines Toolbar.....	41
Drill arrays	
templates	150
Drill bank	
configuring	176, 180
Drill circles	
templates	149
Drill corners	
templates	154
Drill points	
along a contour	
templates.....	153
drill cut parameters.....	154
in centers of contours	
templates.....	151
templates	148
Drill Toolbar.....	36
Drill Tools	147
Dwell	133, 177
Edit Contours Toolbar	41
Edit Relief	
Build Parameter	299
Fade.....	298
Parameters.....	297
Shrink.....	298
Edit Relief Toolbar	297
Editing	
bridges.....	165
entry/exit points	163
Editing points	77
Editing Points	
arc	78
bezier curve.....	78
insert corner tool	78
line segment	77
linearize curve selection.....	78
smooth approximation tool	79

Ellipse		Guidelines.....	12
by major and minor axis	69	creating.....	12, 13
Ellipses		deleting.....	13
by corners	69	hiding	14
by, height and width	68	locking	14
Engrave Toolpath.....	157	moving	13
Engraving		rotating	14
3D toolpaths	135	Hatch angle.....	142
cut parameters	158	Hatch fill	
Entry/exit parameters	134	hatch angle	142
Entry/Exit Points.....	163	hatch vs. Island fill	143
editing.....	163	overlap	142
editing cut parameters.....	164	template.....	142
Entry/exit positions		Hatch Fill Toolpaths	138
displaying	22	Height Control Curve	224
Exiting the software	26	Help	18
Explode Selected Contours	88	Hexagons.....	<i>See</i> polygons
Exporting a design	54	Hold jobs	194
Extend/Trim Toolbar	41	Holes	97
Extending contours	79	Import a Design	54
precise length.....	80	Influence Line.....	224
to boundaries	80	Inlay.....	128, 144
Extrude	230	inlay gap.....	128
extrude dialog	231	Installation	4
extrude example	235	hardware key	3
mesh parameters	234	Island fill	
profile placement.....	233	corner tags	145
rotation parameter.....	233	hatch vs. island fill	143
scale and roatation	232	inlay	144
Extrusions Toolbar.....	37	optimization	144
Faceted.....	264	template.....	145
Fade	298	Island Fill Toolpaths.....	143
Feed rates	177	Jobs	
File Toolbar	27	aborting	195
Files		deleting.....	195
creating	51	Done queue	194
exporting	54	held	194
importing	54	Layers.....	16
Fill cut.....	138	changing a contours layer.....	18
Fill Toolbar	41	Layers Dialog	16
Fillet		Libraries Toolbar	29
multiple fillet	85	License agreement	1
single fillet.....	84	Lines	
Filleting contours	83, 84	drawing	59
Fine Cut	138	editing	77
Flyout Menus	8	extending.....	79
Fractions, converting	127	trimming.....	81
Generating Output	188	Machine drivers	
Grouping Contours	99	active drivers	178
Groups		selecting	189
nesting	121	Mask	287
ungrouping	99	Material Library	181

Measure	56	output parameters	192
Merge.....	87	to a file	193
merge contours	98	to a machine	193
merge selection	87	Output control center.....	193
merge tolerance	87	Output Toolbar	30
merging open contours	87	Overcut	302
Mesh		Overlap	142, 306
apply mesh to relief	264	Passes	132
create surface.....	218	Path Copy	92
extrude mesh example	237	Pentagons	<i>See</i> polygons
spin example.....	227	Plan.....	125
sweep two rails mesh example	245	editing	162
Mirroring a Contour.....	111	plan templates	126
Interactively.....	111	templates	163
mirror horizontal	112	plate	
mirror vertical.....	113	automatic fit	52
Miter	250	Plate.....	51
Modify Relief Toolbar.....	39	displaying.....	22
Move Objects		Plunge.....	177
move tool.....	103	Plunge rate.....	133
moving interactively	100	Points	
moving with precision input center	101	editing	77
Moving		Polyarc.....	59
bridges	166	constructing a line segment.....	60
Multicopy		construction and arc segment	60
arc copy	90	Polygons	
array copy.....	89	drawing	69
copy around an arc	90	Precision Input Center	10
path copy	92	Preferences	19
Multiple Fillet.....	85	dimensions tab	24
Nest Toolbar	40	display tab.....	21
Nesting Contours	119	general tab.....	19
dynamic nest tool	121	grid tab	21
multiple nest tool	122	initialization tab	20
nest objects tool.....	119	ordering tab	25
New files.....	51	relief tab	23
Nose cone optimization	144	start points.....	24
Object Edit Toolbar	31	unit tab	21
Octagons.....	<i>See</i> polygons	view setup tab	22
Offset Contours.....	86	Primitives	262
partial offset	86	Priority.....	190
Offset Toolbar.....	42	Pyramid strategy	
Open contour offset		cut parameters	161
no 3D toolpaths	138	templates	160
relief	137	Pyramid Toolpaths	159
Open Contour Offset Toolpaths.....	136	Quitting	26
Opening a file	53	Radiused corners	83, 84
Ordering options	189	Rectangles	
Ordering toolpaths	170, 171, 173, 322	drawing by corners.....	64
Other Command Icons.....	44	drawing by dimensions	64
Output		Redo	16
2D simulation	289	Redraw	15

Relief	137	revolve dialog.....	214
angles for revolution.....	215	revolve example	217
application method	202	revolve tool	213
add	202	wizard prompts.....	216
merge highest.....	203	Right-click menu	22
merge lowest	204	Rotate	99
replace.....	204	rotate interactively.....	108
subtract.....	203	rotate tool	108
apply bitmap example	285	rotate with precision input center	101
chamfer tool.....	247	Routing offset	
clearing reliefs	270	templates	130
create	199	Routing Offset Toolbar.....	36
delete relief objects.....	270	Routing Offset Toolpaths	126
extrude tool.....	230	Rulers	9
fit relief to plate.....	273	Scale	
inverting	274	reference line.....	106
mesh paramters.....	216	scale 1 dimension	105
mirror	274	scale 2 dimensions	105
modifying and combining.....	269	scale 3 dimensions	105
moving reliefs.....	270	scale tool	104
options.....	200	scale with precision input center	101
constant Height	201	scaling interactively	100
limit to height.....	202	scaling line	106
normal	200	scaling objects	99
scale to height	201	Scale and Rotation.....	232
relief dialog example.....	208	Scale Toolbar.....	40
relief parameters	205	Scaling Reliefs.....	272
angle.....	207	Screen refresh	15
base	206	Scroll bars.....	10
height	205	Setting the Toolpath Order	187
resolution	205	Settings Toolbar	29
revolutions.....	213	Shortcut menu	22
revolve example	217	Shoulder	133
rotating	271	Shrink	298
scaling	271	Simulate 2D.....	289
shape		Simulating Output	169
beveled.....	200	Simulate 2D	169
flat.....	200	Simulate 3D	172
round	199	Simulate Ortho.....	171
smoothing reliefs	273	Single Fillet	84
spin tool	219	Slot cuts	
sweep two rails	239	templates	157
vertical positioning of reliefs	271	Slot Toolpaths	156
wizard prompts	216	Smooth	264
Relief Edit Tools.....	297	Snap.....	11
Relief Toolbar.....	40	snap to center of arc	11
Reliefs		snap to endpoint	11
using bitmaps.....	285	snap to grid.....	11
Rendered View of Toolpaths	322	snap to guideline	11
Return height	160	snap to intersection.....	11
Reverse open Contours	88	snap to midpoint.....	11
Revolve.....	213	snap to nearby contour	11

snap to perpendicular point	11
snap to tangent point.....	12
Snaps Toolbar.....	35
Snapshot	56
Sort method	192
Spin.....	219
height control curve.....	224
influence line.....	224
mesh surface example	227
spin angles.....	221
spin dialog.....	220
spin example.....	226
width control curve.....	225
wizard prompts	223
Spindle speed.....	134, 177
Spiral Fill Toolpaths	146
Stacks.....	222
Stars	<i>See</i> polygons
Start points	
displaying	22
Status Line	9
Step Rough	304
Strategies	
adding.....	162
editing.....	162
removing	162
strategy templates	126, 138
Strategy	125
Strategy order	192
Support Services	5
Sweep Two Rails	239
sweep two rails dialog	240
sweep two rails example.....	243
Templates	53
cut templates.....	126, 135
drill array templates.....	150
drill circle templates	149
drill corner templates.....	154
drill point templates.....	148
drill points along a contour.....	153
drill points in centers of contours templates	151
island fill template.....	145
open contour offset templates	138
plan templates.....	126, 163
plate templates.....	53
pyramid templates	160
routing offset templates	130
slot cut templates	157
strategy templates	126
Text.....	70
convert text objects to curves	73
editing existing text object tool	72
editing text objects	72
Text Toolbar	42
Texture Tool	
parameter dialog.....	294
Texture Tools	291
texture dialog	292
Textures Toolbar	42
Tool changer	
configuring.....	176, 179
Tool library.....	182
custom tool library	184
Tool order.....	191
Toolbars.....	8
Toolpath order.....	192
Toolpaths.....	125
cuts.....	125
direction arrows	22
displaying.....	22
editing groups.....	162
final depth	23
group plan	125
ordering.....	170, 171, 173, 322
plan	125
editing.....	162
toolpath groups.....	125
toolpath strategy.....	125
toolpath width	22
Toolpaths Toolbar	32
Triangles.....	<i>See</i> polygons
Trimming contours	81
Precise Length.....	82
to boundaries	82
Undo.....	16
Units	9, 177
USB	
hardware keys	3
Using a Mask.....	266
Vectorize Bitmap.....	94
Views	8, 327
Weld Toolbar.....	37
Welding contours	
cut by line.....	118
welding options.....	115
Welding Contours	
jigsaw weld	117
Width Control Curve	225
Width of cuts	132
Width of toolpaths	22
workflow	325
Workflow	7
Zoom commands	14
Zoom Toolbar.....	28

